AutoSD STEEL DETAILING

Reference Manual

Version 2015

This manual was published in December, 2014 based on AutoCAD release 14 - 2015 and Bricscad PRO 10 - 15

Trademarks

There are some drawings in the C:\autosd\Bonus folder that you may find useful. The drawing sizes are 24" x 36" and the scale is 1" = 1'-0.

CHARTS.DWG: This drawing has bolt length and bolt values, minimum edge distance, usual gages for angles, minimum weld sizes, pipe dimensions and C and MC dimensions.

FRACTION.DWG: This drawing is used to convert decimals of an inch or of a foot to fractions.

KEYBOARD CARD.DWG: This is a drawing of a keyboard layout showing the location of the custom characters.

WSHAPES.DWG: This is a drawing that shows the dimensions for W4 - W44 and minimum and maximum rows.

13TH EDITION BOLT VALUES.DWG: This drawing list ASD and LRFD bolt values from AISC 13th edition.

These drawings are in the C:\autosd\menus folder. The drawing sizes are 24" x 36" and the scale is 1" = 1'-0.

PULLDOWN MENU DIAGRAM.DWG: This drawing shows the layout of all commands in the AutoSD pulldown menu.

RIBBON MENU DIAGRAM.DWG: This drawing shows the layout of all commands in the AutoSD ribbon menu.

Minimum screen resolution for displaying dialog boxes is 1280 x 960

QUICK START

Read this first.

STEP 1: Installing the program.

If you are installing AutoSD for AutoCAD LT, you will need to install one of the Lisp Enablers. Go to www.autosd.com/support.htm for a link to find where to purchase this product.

Updating an older version of AutoSD Steel Detailing:
You should uninstall the current copy of AutoSD Steel Detailing first. Uninstalling AutoSD will not remove any drawings or customer configurations that you have created.
Windows XP: From the task bar pick Start, Settings and Control Panel. Double click on Add or Remove Programs in the Control window.
Windows 7: From the task bar pick Start and Control Panel. Click on Uninstall a program in the Control Panel window.
Windows 8 & 8.1: Change to the desktop screen. Right click on the Windows icon and pick Control Panel. Click on Uninstall a program in the Control Panel window.

Select AutoSD Steel Detailing XX, where XX is the version number and pick Uninstall.

Note: If you have customized your autosd.dwt template file located in your AutoSD\support folder you will need to save it to another folder or disk or rename it before un-installing AutoSD. After installing AutoSD, copy your old autosd.dwt template file back into your AutoSD\support folder.
All files will not be removed. Some files and folders will remain. Do not delete the remaining files and folders. Install AutoSD over the existing files.

Insert the CD-ROM into your CD-ROM drive and if the installation does not start, choose START, RUN. Type X:\AUTOSD_SETUP where X is your CD-ROM drive letter. Follow the on-screen prompts.

STEP 2: Installing the hardware lock.

After Setup is finished:
Attach the USB hardware lock to a USB port.

STEP 3: Configure AutoCAD or BricsCAD.

Start AutoCAD or BricsCAD. See appendix A for complete instructions on setting the support file search path, template location and loading the menu. If you are using AutoCAD 14, close AutoCAD and reopen it after setting the support file search path and before loading the menu.

Note: If you are updating an older version of AutoSD you must unload the AutoSD menu and reload it. See appendix A for instructions on loading the AutoSD menu.

STEP 4: Configure AutoSD.

After configuring AutoCAD and loading the AutoSD menu, select the AutoSD pulldown menu and Customer Configuration. Select Detail variables. Select Unnamed and pick Edit. Change the settings in the dialog boxes as needed and select OK. Select Color list. Select AutoSD from the list of names and pick Edit. Use the dialog box to set each color by picking on the color of the item to be changed. Colors for weld symbols are set in the Weld Menu dialog box. Pick OK to close the Setting Colors dialog box. You can rename the file names and add more configurations if needed. Pick OK again to exit the configuration.

STEP 5: Starting AutoSD.

Start AutoCAD or BricsCAD. If you are using AutoCAD LT you need to use the Lisp enabler desktop icon to start AutoCAD LT.

Select File > NEW. Select "Use a Template" and select the autosd.dwt template. It is in your C:\autosd\support folder.
Pick AutoSD > Format > SETUP or AutoSD I > Utilities > SETUP.
Select a configuration from each of the four boxes.
Select AutoSD for Detail Variables, Unnamed - 24x36 for Shop bill, None for Material list and AutoSD for Color list and pick OK.
In the Setup box, accept the defaults which are: Architect for Unit type 1 = 1’ for Drawing scale, D - 24 x 36 for drawing size, horizontal for paper size and AutoSD.
Pick OK and the drawing is created.
Pick File > Save and save the drawing.
You can start detailing now.

A folder called C:\AUTOSD\CUSTOMER is created by the installation program. Where C: is the drive you installed AutoSD on and AUTOSD is the name of the folder you chose to install AutoSD in. You can use the shop bill drawings in this folder, (CVSPro.dwg, 24x36BOM.dwg and 11x17BOM.dwg), to create your own shop bill. Open the drawing, change the name in the title box, save and close the drawing.

Uninstalling AutoSD.

Open the Control Panel and pick Uninstall programs. Scroll down the list of programs until you find AutoSD Steel Detailing 2015. Select it and pick uninstall.

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Chapter 1 - Introduction

Welcome to AutoSD Steel Detailing.

AutoSD Steel Detailing, (AutoSD), is a steel detailing program that is designed to increase the production of the draftsperson and act as an aid for the checker. It is assumed that you have a working knowledge of steel detailing and AutoCAD or BricsCAD It is not designed to replace the draftsperson but is an enhancement. The draftsperson is in control of the detailing at all times. AutoSD will do most of the calculations and all of the drawing normally done by the draftsperson in detailing beams, columns, bracing, gusset plates, stairs, handrails, ladders, erection plans, sections, details and more.

MENUS

AutoSD comes with two menus, autosd.mnu and autosd.cuix. All versions of AutoCAD will load the *.mnu file but you need AutoCAD 2010 or later to load the *.cuix file. The difference between the two menus is the *.cuix file contains a ribbon menu and the *.mnu file does not. The ribbon menu has two tabs, AutoSD I and AutoSD II. In this reference manual you will see [AutoSD or AutoSD I]. “AutoSD” refers to the pulldown menu. “AutoSD I” or “AutoSD II” refers to the ribbon menu tab. You can use the ribbon menu exclusively or just the pulldown menu. Toolbars can be used with either menu. The AutoSD menu is an add-on to the standard AutoCAD menu. Ribbon menus can be turned on and off with the commands RIBBON and RIBBONCLOSE. Pulldown menus can be turned on and off with the command MENUBAR. Set the value to 1 to turn it on and to 0 to turn it off.

FT.IISS or II.SS

You will see this in most of the input prompts. It is the standard method of entering dimensions. If the current dimensional unit format is set to architectural the FT.IISS format will be used. If it is set to fractional the II.SS format will be used.

Where:
FT = number of feet
II = number of inches
SS = number of sixteenths

Using the **FT.IISS** format:
1'- 3 1/2" would be entered as 1.0308
6'- 10 15/16" would be entered as 6.1015
4'-0" would be entered as 4
3 1/2" would be entered as .0308

Using the **II.SS** format:
15 1/2" would be entered as 15.08
82 15/16" would be entered as 82.15
48” would be entered as 48
1/2" would be entered as .08

This manual uses the FT.IISS format in the examples.

POSITIVE AND NEGATIVE

Many programs provide you with slide drawings or images in dialog boxes for user input. Positive input will go in the direction shown in the slide; negative input will go in the opposite direction from that shown in the slide.

FT-II*SS

You will see this in most of the input prompts when you set up a drawing for CVSpro. It is the standard method of entering dimensions for CVSpro. If the current dimensional unit format is set to architectural the FT-II*SS format will be used.

Where:
FT = feet II = inches SS = sixteenths

Using the **FT-II*SS** format:
1’- 3 1/2" would be entered as 1-3*12
6’- 10 15/16" would be entered as 6-10*15
4'-0" would be entered as 4-
5 1/2" would be entered as 5*12
3" would be entered as 3
1/2" would be entered as *12

Feet are entered followed by a minus sign. 2’-0 = 2-
Inches are entered as a whole number. 3” = 3
Fractions are entered preceded by an asterisk. 1/2” = *12
Fractions are as follows:

1/16 = *1	5/16 = *5	9/16 = *9	13/16 = *13
1/8 = *18	3/8 = *38	5/8 = *58	7/8 = *78
3/16 = *3	7/16 = *7	11/16 = *11	15/16 = *15
1/4 = *14	1/2 = *12	3/4 = *34	

ENTER

Most AutoCAD commands will repeat themselves when you press the ENTER key. This saves time by not having to go back to the menu to pick the command each time when you are repeating the same command over and over again such as the copy or erase command. AutoSD takes advantage of this same time saver in many of its commands such as editing and dimensioning, and many programs.

FILE NAMES

AutoSD uses several different file name extensions. All of these, except for the .cfg, bak, dat and .sld files, are saved in the folder of the current drawing. The extensions and their uses are as follows:

.blt Bolt list extracted from your drawing.

.bak Configuration bakup files.

File name	Folder	Purpose
CUSTBOM.BAK	C:\autosd	Shop bill program.
CUSTBOMM. BAK	C:\autosd	Metric shop bill program.
CUSTCLR. BAK	C:\autosd	Color list.
CUSTMAT. BAK	C:\autosd	Material list.
CUSTMATM. BAK	C:\autosd	Metric material list.
CUSTVAR. BAK	C:\autosd	Detail variable configuration.
CUSTVARM. BAK	C:\autosd	Metric detail variable configuration.
SCHMAKER. BAK	C:\autosd	Schedule maker
SDS2CLR. BAK	C:\autosd	SDS/2 colors
SETUP12. BAK	C:\autosd	Custom setup settings
WELD. BAK	C:\autosd\welds	Weld symbols

.cfg Configuration files.

File name	Folder	Purpose
ASD_DATA.CFG	C:\autosd	List of data files for each shape.
CUSTBOM.CFG	C:\autosd	Shop bill program.
CUSTBOMM.CFG	C:\autosd	Metric shop bill program.
CUSTCLR.CFG	C:\autosd	Color list.
CUSTMAT.CFG	C:\autosd	Material list.
CUSTMATM.CFG	C:\autosd	Metric material list.
CUSTVAR.CFG	C:\autosd	Detail variable configuration.
CUSTVARM.CFG	C:\autosd	Metric detail variable configuration.
DATLIST.CFG	C:\autosd	Stores values for dialog boxes.
DATLISTM.CFG	C:\autosd	Stores metric values for dialog boxes.
SCHMAKER.CFG	C:\autosd	Schedule maker
SDS2CLR.CFG	C:\autosd	SDS/2 colors
SETUP12.CFG	C:\autosd	Custom setup settings
WELD.CFG	C:\autosd\welds	Weld symbol configuration

.col Column files for the column program.

.csv Shop bill extracted for E.J.E. material manager

.dat Data files for shapes and to store defaults for some programs.

.dwg AutoCAD drawing.

.err Omissions from the .imp or .kss file when extracting bill of material.

.gus Gusset plate files for the bracing program.

.imp Shop bill extracted for E.J.E. material manager

.kss Shop bill extracted to the kiss format for Fabtrol.

.lst BMCONN.LST Used to store beam end connections used in beam to beam connection matching.

.mom Moment end plate files for a beam connection.

.nc DSTV format cnc file. Extension can be changed in customer configurations.

.pl List of cap and base plates. File name is BASECAP.PL.

.sbl Shop bill extracted for viewing and printing using the Extract program.

.sld	Slides in the SLIDES folder of the drawings stored in the BLOCKS folder.
.tbl	SUBMARK.TBL. For sub marks that carry through.

If a problem occurs after editing a customer configuration file you can delete the associated cfg file and rename the bak for that configuration to cfg.

ROTATION

Some programs give you the option of rotation. All angles are measured in a counter clockwise direction with right being 0 degrees. Up is 90 degrees, left is 180 degrees and down is 270 degrees. Do not change this setting or the programs will not draw the pictures correctly.

Angles are entered and displayed in degrees by default. You can, however, change the units of a drawing so degrees can be entered in any of the options offered by AutoCAD.

When drawing the end view of a steel shape, such as a wide flange, a rotation of 0 degrees is in reference to the flange and the web will be drawn up and down. This is the position you would normally view the end of a beam.

SCALE

There are many different scales provided that you can set your drawing up to. However, when detailing beams, columns, or stairs you should use a scale of 1" = 1', (1:12 for metric). This is the standard scale for detailing steel and should be used for all detail sheets. A feature of AutoSD not found in most other CAD detailing software is that of being able to change the scale you are drawing to. Even though you have set your drawing up to 1" = 1' as the base scale you can use the "[AutoSD > Modify or AutoSD I > Change] > Scale of Drawing" command and draw sections and details to a different scale. See chapter 3 for a list of some of the programs and commands affected by the change drawing scale command.

The flange and web thickness on thin members will be exaggerated. Flanges will be drawn a minimum of 3/4" and webs a minimum of 1/2".

CONVERTING OTHER DRAWINGS TO AutoSD

If you have a drawing that was created by software other than AutoSD and want to edit the drawing with AutoSD certain variables need to be set in the drawing for AutoSD to work correctly. Set up a new drawing using AutoSD and insert your old drawing as a block into it and explode the inserted block.

If you have a DWG file created by SDS/2, SteelCAD, CVSpro or Xsteel that was imported into AutoCAD from a dxf file you can convert it to AutoSD by opening the drawing and running [AutoSD > Format or AutoSD I > Utilities] > Setup. Select the drawing scale and paper size you want to use and select CVSpro, SDS/2, SteelCAD or Xsteel and pick OK.

If you have a DXF file created by SDS/2, CVSpro or Xsteel you can import it into AutoSD. If you have an older version of SDS/2 that drew holes with an open circle in the AutoCAD DXF

file then you must select the color used for the holes and it must be a different color from all other lines, otherwise select “Ignore” for the color choices.

Start AutoCAD and start a new drawing using the autosd.dwt template. Do **NOT** run setup. From the [AutoSD > Tools or AutoSD I > Utilities] menu select **DXF into AutoSD** and then select *Single* or *Multiple* **SDS/2 DXF file**, **CVSpro DXF file** or **Xsteel DXF file**. (*Multiple* is NOT available for AutoCAD LT). For a single file select the dxf file from the dialog box. For multiple files select a single file in the folder that contains all of the files to import. Only files with a “.dxf” extension will be imported.

After importing the dxf file the drawing is moved so the lower left corner of the border is at the origin, (0, 0). The **SDS/2 DXF** file is scaled up by a factor of 12 to make the scale 1”=1’-0 and the fractions are changed back to a stacked fraction. **Xsteel DXF** files are scaled to fit the margins set by the setup program for a 24” x 36” sheet.

SDS OUT

Dwg to SDS/2 Dxf

Pulldown: AutoSD > Tools > Dwg to SDS/2 Dxf
Ribbon: AutoSD I > Utilities

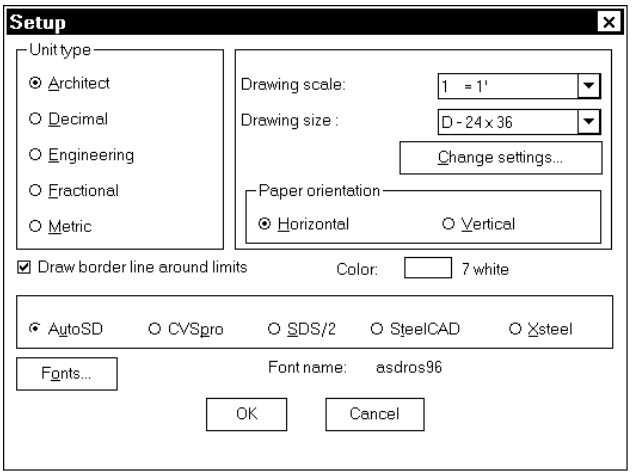
This command will convert your drawing and create a dxf file of the same name as the dwg file and save it to the same folder. All attributes will be replaced with text. Some information in the field bolts will be lost after converting. If you want to save this information you should make a backup copy of the dwg file to another folder before converting it.

STARTING A NEW DRAWING...

Starting a new drawing that is compatible in appearance to drawings created by SDS/2, STEELCAD, CVSpro or Xsteel.

Select Files and New. Select "Use a Template". In the "Select a Template" window select the autosd.dwt template and pick OPEN. If the templates are not in the window they can be found in your autosd\support folder.

Pick [AutoSD > Format or AutoSD I > Utilities] > Setup. Select SDS/2, SteelCAD, CVSpro or Xsteel option at the bottom of the setup dialog box.



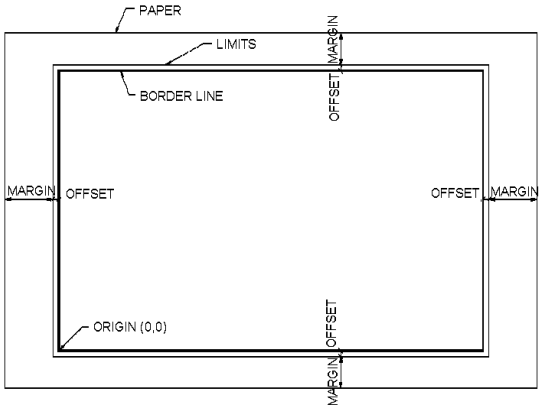
SETTING UP

Pulldown: AutoSD > Format > Setup
Ribbon: AutoSD I > Utilities

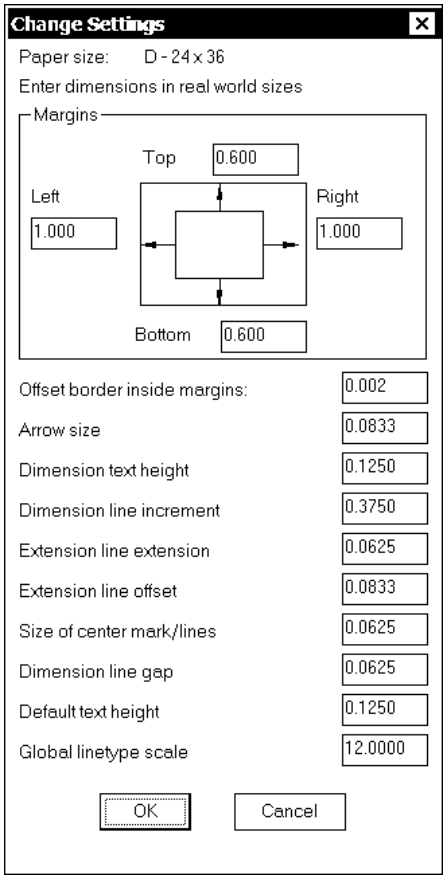
When you start a new drawing the first thing you have to do is set the scale, units and paper size. A dialog box is used for

this purpose. Pick [AutoSD > Format or AutoSD I > Utilities] > Setup. Your next selection will be for the units. You may pick "ARCHITECTURAL", "DECIMAL", “ENGINEERING”, FRACTIONAL” or "METRIC”. Dimensions in architectural and fractional units will be rounded to the nearest 1/16th, decimal units will be rounded to 2 decimal places and metric units to the nearest millimeter. After selecting units you must pick the scale and the paper size. All detailing should be to 1"=1' scale, or 1:12 for metric. If you need a scale not listed, select "Type it". After picking the OK button you will be prompted to "Enter scale in inches per foot". If you want a scale of 7/32" = 1'-0" then divide 7 by 32. You would enter .2188 as the scale. The Plot scale would be the reciprocal x 12 = 54.8571. For a list of the scales and Plot scales provided see page 13.

When you pick OK the drawing is set up. One border is drawn around the sheet if the border option is checked. Save the drawing and then you may begin drawing.
Three layers are created for drawing, ASD_DRAW which is used for object lines, dimensions and text and ASD_SECT which is used for section symbols and ASD_MARK which is used for main marks. The color of layer ASD_DRAW determines the color of text. You can change the colors that are used by default by editing the colors with the “Customer Configuration” command. See page 14 for more information.



To change the drawing area and limits of a selected paper size pick the “Change settings” button. Enter the margin sizes in real world distances. The offset of the border inside the margins should be greater than 0. If 0 is used the border may not plot if the drawing is plotted using limits. The lower left corner of the border will be placed at the origin of 0,0 and the border will be inside the limits by the amount of the offset.
The border and drawing limits are smaller than the paper size. The chart below lists the paper sizes and the default drawing area.

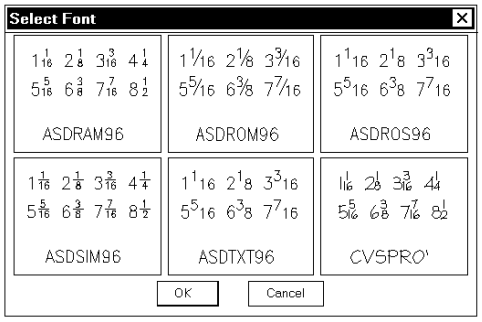


ARCHITECTURAL and DECIMAL		METRIC	
PAPER SIZE	LIMITS	PAPER SIZE	LIMITS
A-8.5x11	7.5"x10"	A-8.5x11	190x253
B- 11x17	9.8"x16"	B- 11x17	253x402
C- 22x34	20.8"x32"	C- 18x24	427x560

C- 18x24	16.8"x22"	C- 22x34	529x814
D- 24x36	22.8"x34"	D- 24x36	580x864
D- 30x42	28.8"x40"	D- 30x42	732x1016
E- 36x48	34.8"x46"	E- 36x48	884x1169
		A4-210x297	190x247
		A3-297x420	267x370
		A2-420x594	390x544
		A1-594x841	594x791
		A0-841x1189	801x1139
		A4-240x330	210x300
		A3-330x450	300x420
		A2-450x625	410x595
		A1-625x880	585x850
		A0-880x1230	440x1200

OTHER You enter the paper size and margins

Some dimension variables are stored in the Setup dialog box. If you want to change the arrow size, dimension text height, dimension line increment, extension line extension, extension line offset, size of center marks/lines, dimension line gap and default text height that will be set when you create a new drawing you must change the values here. If you want to change any of these values after the drawing is setup you change them in the Dimension Style dialog box for the AUTOSD dimension style.



To change the font, pick the Font button. Select one of the seven fonts shown in the icon menu. The name of the font is listed at the bottom of each slide and will be shown to the right of the Font button. The font selected will become the default font used when you set up a new drawing. A different font may be saved with AutoSD, SDS/2, SteelCAD and Xsteel.

TEXT HEIGHT and PLOT SCALES

In order for the text to appear the same size when plotted using different drawing scales the text height must change with each scale. AutoSD will make this change automatically when you set up your drawing. The text will be 1/8" in height when plotted. You can, however, change this setting each time you use the text command. The chart below shows the correct text height and plot scale for each drawing scale.

ARCHITECT			DECIMAL			METRIC		
SCALE	**HEIGHT**	**PLOT SCALE**	**SCALE**	**HEIGHT**	**PLOT SCALE**	**SCALE**	**HEIGHT**	**PLOT SCALE**
1/16"=1'	24	192.0	1"= 2000'	3000	24000.0	1:5000	15875	5000.0
3/32"=1'	16	128.0	1"= 1000'	1500	12000.0	1:4000	12700	4000.0

1/8" =1'	12	96.0	1"= 500'	750	6000.0	1:2500	7938	2500.0
5/32"=1'	9.6	76.8	1"= 250'	375	3000.0	1:1250	3969	1250.0
3/16"=1'	8	64.0	1"= 200'	300	2400.0	1:500	1588	500.0
1/4" =1'	6	48.0	1"= 100'	150	1200.0	1:300	952	300.0
5/16"=1'	4.8	38.4	1"= 80'	120	960.0	1:200	635	200.0
3/8" =1'	4	32.0	1"= 60'	90	720.0	1:100	318	100.0
1/2" =1'	3	24.0	1"= 40'	60	480.0	1:75	238	75.0
5/8" =1'	2.4	19.2	1"= 30'	45	360.0	1:50	159	50.0
3/4" =1'	2	16.0	1"= 20'	30	240.0	1:20	64	20.0
1" =1'	1.5	12.0	1"= 10'	15	120.0	1:12	38	12.0

ARCHITECT			DECIMAL			METRIC		
SCALE	**HEIGHT**	**PLOT SCALE**	**SCALE**	**HEIGHT**	**PLOT SCALE**	**SCALE**	**HEIGHT**	**PLOT SCALE**
1 1/4"=1'	1	9.6	1"= 1'	1.5	12.0	1:10	32	10.0
1 1/2"=1'	1	8.0	FULL	0.125	1.0	1:5	16	5.0
3" =1'	.5	4.0				FULL	3	1.0
FULL	0.125	1.0						

You can also change the height of the text and dimensions by selecting [AutoSD > Format or AutoSD I > Utilities] > Text size and the size you want to use. 5 is normal (1/8"), 1 is half size and 10 is double size.

FOLDERS

Folders must exist before you can put something in them.

It is a good idea that you use separate folders for each job to store your drawings. This will keep your hard drive well organized. When you start a new drawing at the main menu give the path to the folder where you want to save the drawing. If you want drawing number 1 to be saved in the folder C:\SSS\9501, where "SSS" is the customer and "9501" is your job number, the correct path for drawing number one is C:\SSS\9501\1. This will keep the drawings for one job and customer separate from those of another.

CUSTOMER CONFIGURATION PROGRAM

Before you begin using the program you should first create a customer folder and a job sub folder to be used to save drawing files. You can use Windows Explorer to do this. The root folder should be the name or abbreviation of the customer you are detailing for. A sub folder for each job will be set up inside this folder.

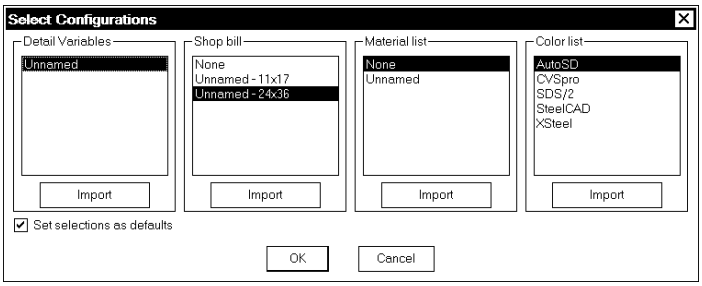
Example:	C:\SSS	The customer folder SSS
	C:\SSS\9501	The job sub folder 9501

Warning - AutoCAD 14 only: All folder names in the path must be eight characters or less and must not contain any spaces. Folder names must conform to DOS restrictions. This is a

requirement of the extract and import programs for processing multiple drawings. Likewise, drawing file names must also conform to the DOS restrictions of 8 characters or less.

Refer to Windows Help on how to use Windows explorer.

This example is for a customer called **S**uperior **S**teel **S**ervices, job number 9501. When you start a new drawing for sheet number one, save it to the job folder created for this customer. In this case it will be C:\SSS\9501 for job 9501 for Superior Steel Services on drive C:. This keeps your hard drive well organized and enables you to have more than one drawing with the same name by keeping them in separate sub folders for each job. When AutoSD is installed a folder called CUSTOMER is created as a sub folder of your AutoSD and has a shop bill drawing file in it called 24x36BOM.dwg and 11x17BOM.dwg. You can edit and use these drawings to get started. This is intended as a temporary customer folder that you can use to save drawings in until you create your own.



The first time you select to detail something, a dialog box will pop up for you to select the detail variables configuration, shop bill configuration, material list and colors. The default file name for the detail configuration is "Unnamed". The one for shop bill and material list is "None". The file "None" is not a configuration but is a selection you can make if you do not want the program to fill out a bill of materials or use a materials list for standard marks. The default name for Colors is AutoSD.

This dialog box allows you to select any combination of detail variables, shop bill, material list and color configurations. This allows you to have several paper sizes for one job and several different customers.

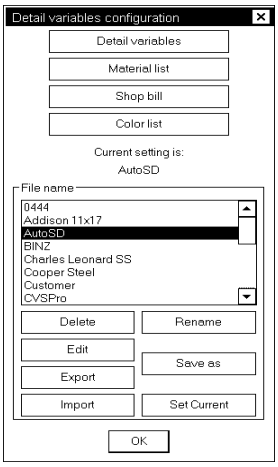
If you had an older version that used the detvar.cfg, sbtext.cfg and material.tbl files you can import them with the Import buttons. You can also import individual configurations from other custVAR.cfg, custBOM.cfg, custMAT.cfg and custCLR.cfg files. If you need to edit a file or set a different one current you will use "Customer Configuration" to do this.

After selecting the configurations to use for this drawing pick OK. The names selected will be stored in the drawing in a block called asdcust. The next time the drawing is opened and edited you will not be asked to select the configurations to use unless the names stored in the drawing do not match any configuration files. This can happen if the file names were changed.

EDITING A CONFIGURATION

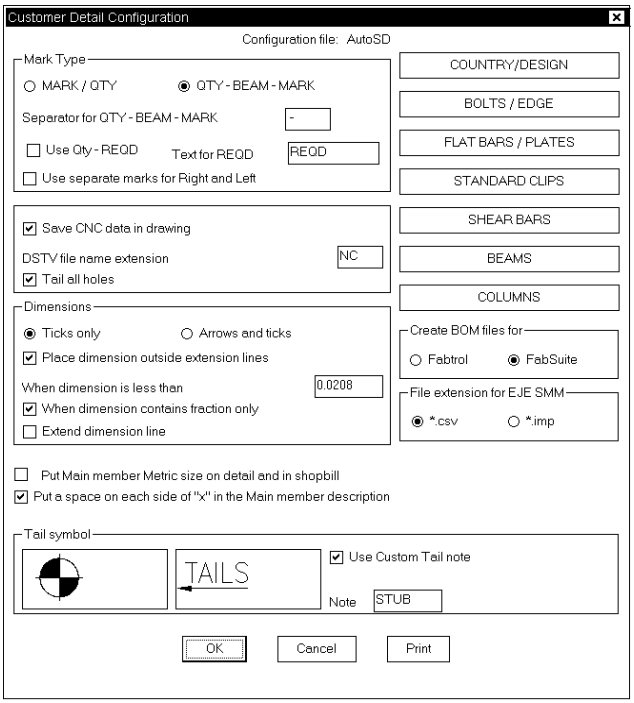
Pulldown: AutoSD > Customer Configuration
Ribbon: AutoSD I > Shop bill

Pick one of the buttons in the top section to display a list of configurations under "File name"

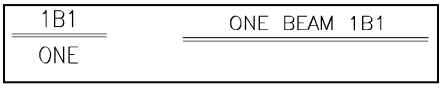


The first time you open the configuration box and pick "Detail variables" you will have one file in it called "Unnamed". You can edit and use this file. You can rename it or copy it with the "Save as" button and make more files. If you have more than one file you can delete them but you cannot delete the last one. There will always be at least one file. If you have an older version that used the detvar.cfg file you can import those files and use them. Older versions had separate detvar.cfg files saved in each job folder. This version has all of the configurations stored in one file called custvar.cfg located in your autosd folder. The metric version is called custvarm.cfg. You can also import and export individual configurations. Select the configuration you want to use and pick "Set Current" to save the setting in the drawing.

Select a configuration file and pick the EDIT button. The Customer Detail Configuration box pops up. If you did not purchase the beam program, the Columns and Standard Clips buttons will be disabled.



Marking style for main marks



Mark/Qty Qty - Beam - Mark

You can enter the character that you want to use as a separator between quantity and descriptor and between the descriptor and mark. The default separator is " - ".

Putting a check in the box for "Use QTY – REQD" will put the text shown in the "Text for REQD" box after the quantity separated by the separator character.

Putting a check in the box for "Use separate marks for Right and Left" will cause the program to put one main mark under the piece for the right mark and one for the left mark.

Save CNC data in drawing: Check this box to save CNC data for holes and burns. Additional questions for holes will be asked when detailing if this box is checked. If this box is checked tail dimensions must be from end of material, not from the clip. Enter the file name extension you want to use for DSTV CNC files. Do not include a "." here.

Tail all holes: Check this box to put tail dimensions to every hole. Uncheck it to put tail dimension to the first hole only in a group.

Dimensions: Select either "Ticks Only" or "Arrows and ticks". If Arrows and ticks is selected ticks will be used for dimensions that are smaller than 2.5 times the arrow size. Arrows will be used for dimensions that are 2.5 times the arrow size and larger.

Place dimensions outside extension lines: When dimensions are less than:
If this box is checked, dimensions smaller than the length given will be placed outside of the extension lines.

When dimension contains fraction only: Put a check in this box if you want 1” and 2” dimensions to be centered.

Extend dimension line: Put a check in this box if you want these dimensions to have the dimension line extended under them when they are offset outside of the extension line.

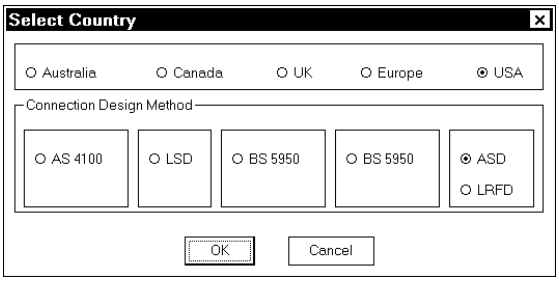
Put Main member metric size on detail and in shop bill: If you have the drawing set up as architectural units, put a check in this box if you need the metric size of the main members shown in addition to the imperial size. If you have the drawing set up as metric units, put a check in this box if you need the imperial size of the main members shown in addition to the metric size.

Tail Symbol: Select the target or "TAIL" symbol to be placed at the left end of the detail or check the box "Use Custom Tail note" to change the word "TAILS" in the tail symbol to the one you type into the edit box to the right of "Note".

Print: Print a hard copy of all of the configuration settings.

Create BOM files for: The options are Fabtrol and FabSuite. Each one imports a *.kss file but each uses a different shape designations. Choose the material management program you are using.
File extension for EJE SMM: The options are *.csv for the newer ASCII delimited file format and *.imp for the older fixed field file format. Choose the extension for the version of SMM that you are using.

COUNTRY

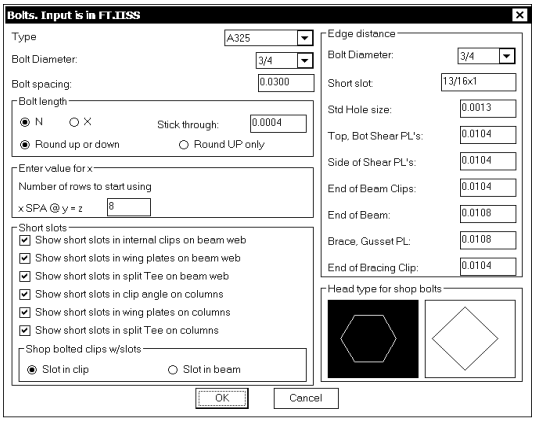


The country you select will determine which data base will be used for shapes and which design method will be used for connections. If you choose any country other than USA your drawing must be set up using metric units.

BOLTS

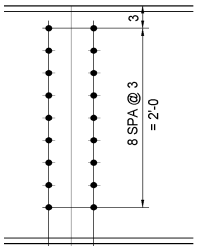
This box is for setting the type, diameter, bolt spacing, bolt length, when to use X SPA @ Y = Z, (where X is the number of rows), graphics for the bolt head and edge distances for each bolt diameter. The type, diameter and bolt spacing are also available on a per detail basis.

Bolt length:
"N" and "X": "N" is for threads included, "X" is for threads excluded from the shear plane.
Stick through: The length of bolt to extend past the face of the nut.
Round up or down: Rounds the length of the bolt to the nearest 1/4" increment but may make the actual stick through 1/16" less. Variance for actual stick through is -1/16", +1/8".
Round UP only: Rounds the length of the bolt UP to the nearest 1/4" increment. Variance for actual stick through is -0, +3/16".



Short slots: Putting a check in any of these boxes will draw a short slot in the connection material. Xdata for cnc will be saved in the piece mark.
Show short slots in internal clips on beam web:
Show short slots in wing plates on beam web:
Show short slots in split Tee on beam web:
Show short slots in clips angle on columns:
Show short slots in wing plates on columns:
Show short slots in split Tee on columns:

Shop bolted clips w/slots:
Putting a check in "Slot in beam" will draw a slot in addition to the bolt head. The slot will contain the xdata for cnc if cnc data is being saved.

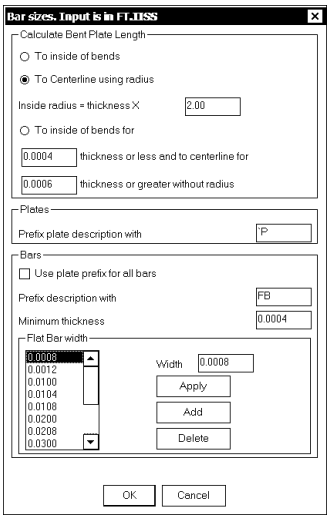


The default for X in number of rows to start using "X SPA @ Y = Z" is 8. Set this value to a very large number if you do not want the hole spacing dimensioned as shown to the left.

Use the Bolt diameter list on the left side of the box to set the standard bolt size to use for connections. Use the Bolt diameter list on the right side of the box under "Edge distance" to change the edge distance for each bolt size. To edit an edge distance select the bolt size and pick the edit box for the edge distance you want to change.
Make the change in the box. The change is saved when you pick OK. If you pick Cancel your changes will not be saved.

FLAT BARS & PLATES

This dialog box is for configuring how the length of bent plates are calculated and for configuring the size of material you want to designate as Flat Bars.
Calculate Bent Plate Length: Options are to inside of bend regardless of thickness, to centerline using a radius based on the thickness and to inside of bend for thin plates and to centerline for thicker plates.



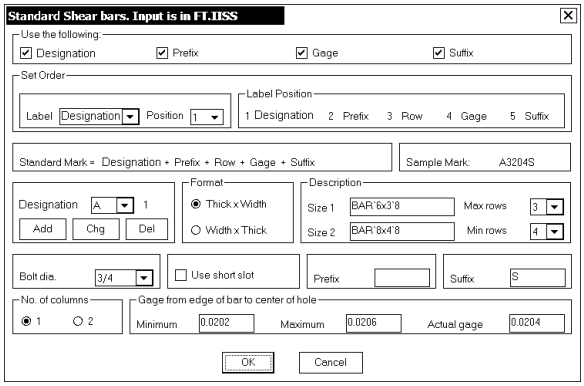
Plates: This is for specifying the prefix used for plates. The default is "`P". This will be shown as "PL" in the drawing.

Bars: If you want all bars to use the prefix for plates, put a check in the box at the top of the "Bars" section or change the prefix for bars to the one shown for plates.

Warning! If you change the prefix for bars to `P you should set the "Format" in "Shear Bars" to "Thick x Width".

The "Width" box lists all of the widths that will be described using the prefix in the "Prefix descriptions with" box. Any plate that is the minimum thickness listed or thicker and whose width is in the "Width" list will be prefixed with the prefix shown. You can add, change and delete widths, set the minimum thickness and change the prefix. To change a width, select the width you want to change, make the change in the box to the right of the word "Width" and pick "Apply".

STANDARD SHEAR BARS



This box is for setting the marks for standard shear bars. The top row contains options you can turn on or off by checking or un-checking the option. Designation and Gage are global settings and apply to all marks. For example, you cannot have some marks with a gage and some marks without a gage. Prefix and Suffix, however, are applied separately to each bar. The row below these options labeled "Set Order" is for arranging the order of the components of the mark. To change the order, select the label you want to move. The current position will be shown in the "Position" drop down list and in the "Label Position" box to the right. Select a new position and the label will be swapped with the one currently in that position. The row below "Set Order" shows how the mark is constructed and a sample of what the mark will look like.

Checking "Designation" will include the text that is shown in the designation pop down list in the mark The text shown in the list can be changed using the "Add", "Chg" and "Del" buttons. If you delete a designation you will be deleting a description. If you add a designation you will be adding a description. The description for the new designation will be copied from the current description. Different descriptions can have the same designation. The number to the right of the designation indicates which description in the designation list you are editing. When editing an existing mark or creating a new mark you start with the designation list. The designation does not have to be part of the mark but is used to separate the descriptions from one another.

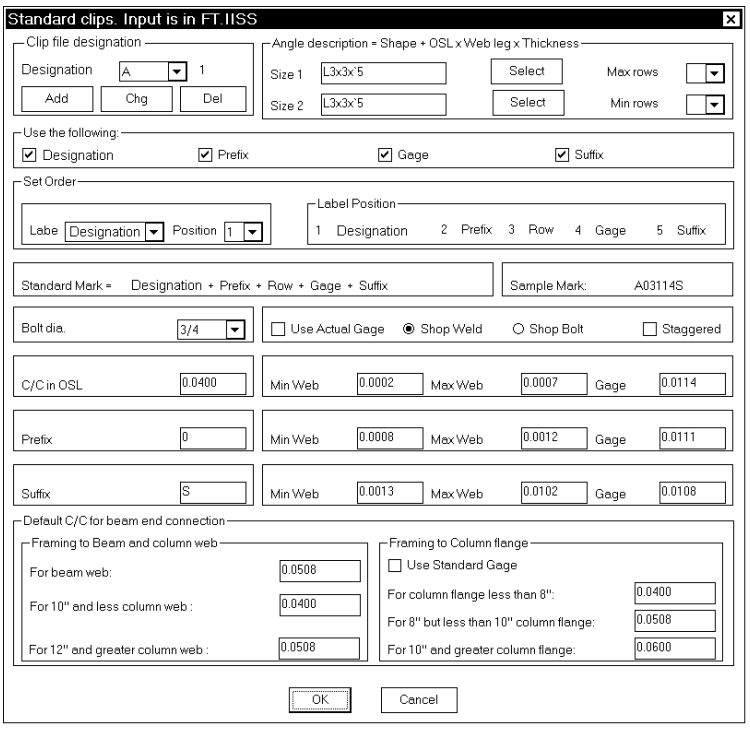
"Format" is where you select which you want listed first, the thickness or the width. This setting is for all descriptions of standard shear bars and the descriptions of flat bars listed in the "FLAT BAR" configuration.

The bar or plate size shown in the "Description" box will be used in the shop bill when a standard mark is used and will also be used to draw the bar to scale. The bar size shown for "Size 1" will be used up to the maximum rows selected. The bar size shown for "Size 2" will be used starting with the minimum rows selected. If the "Max rows" is blank the bar size shown for "Size 1" will be used for all rows and "Size 2" will be disabled.

The values that determine which mark will be used are "Bolt dia", "No. of Columns", "Gage from edge of bar to center of hole" and if the bar has slots or not. A bar with slots will be selected ONLY if the "Use short slot" is checked and “Use standard shear bar with slots” in the “BEAMS” dialog box is checked. If you want the holes drawn as slots you need to check the options "Show short slots in wing plates on beams" and/or "Show short slots in wing plates on columns" in the "BOLTS/EDGE" dialog box. If all of your standard shear bars use short slots and none have round holes and you want to use your standard bars in combination with round holes in wing plates you will need to make a copy of each description of standard bars, one with short slots and one without short slots.

Checking "Prefix" will include the text shown in the "Prefix" edit box as part of the mark. Checking "Gage" will include the listed gage for a given web thickness. The gage will be in the format of inches and 16th's where 2 1/4 is 204. If you want the gage to be represented by a single digit enter it as .0001 for 1 or .0005 for 5.
Checking "Suffix" will include the text shown in the "Suffix" edit box as part of the mark.

STANDARD CLIPS



This box is for creating the marks for standard bolted and welded clips, setting the angle descriptions and default c/c for beam end connections. Start in the upper left corner with the designation. The text shown in the designation list can be changed using the "Add", "Chg" and "Del" buttons. If you delete a designation you will be deleting a description. If you add a designation you will be adding a description. The description for the new designation will be copied from the current description. Different descriptions can have the same designation. The number to the right of the designation

indicates which description in the designation list you are editing. The designation does not have to be part of the mark but is used to separate the descriptions from one another. The values that determine which mark will be used in the beam program are "Bolt dia", "C/C" in OSL", "Shop Weld", "Shop Bolt" and "Staggered".

The angle size shown for "Size 1" will be used up to the maximum rows selected. The angle size shown for "Size 2" will be used starting with the minimum rows selected. If the "Max rows" is blank, the angle size shown for "Size 1" will be used for all rows and "Size 2" will be disabled. The angle size configured will be used in the shop bill when a standard mark is used and will also be used to draw the clip to scale. The first leg size will be the outstanding leg. The second leg size will be placed against the web of the beam. You can enter the angle description manually or pick "Select" and choose an angle from the list that pops up.

The next row contains options you can turn on or off by checking or un-checking the option. Designation and Gage are global settings and apply to all marks. For example, you cannot have some marks with a gage and some marks without a gage. Prefix and Suffix, however, are applied separately to each angle. The row below these options labeled "Set Order" is for arranging the order of the components of the mark. To change the order select the label you want to move. The current position will be shown in the "Position" drop down list and in the "Label Position" box to the right. Select a new position and the label will be swapped with the one currently in that position. The row below "Set Order" shows how the mark is constructed and a sample of what the mark will look like for a 3 row clip.

Checking "Designation" will include the text that is shown in the designation pop down list in the mark. Checking "Prefix" will include the text shown in the "Prefix" edit box as part of the mark. Checking "Gage" will include the listed gage for a given web thickness. The gage will be in the format of inches and 16th's where 1 7/8 is 114. If you want the gage to be represented by a single digit enter it as .0001 for 1 or .0005 for 5. If you are not using slots in the clip or your mark requires the actual gage then check the box by "Use Actual Gage". Checking "Suffix" will include the text shown in the "Suffix" edit box as part of the mark.

If the clip is shop welded, pick "Shop weld". If the clip is shop bolted, pick "Shop Bolt". If the shop bolts are staggered, check the box by "Staggered".

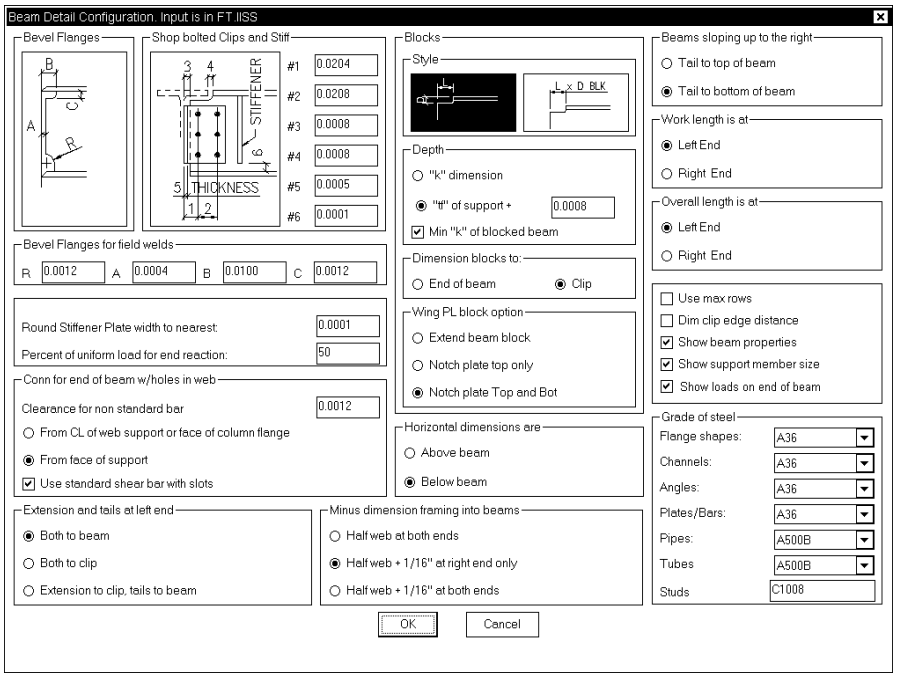
The last items are the Default C/C for beam end connections. These settings should match one of the "C/C in OSL" that you set for one of the clips. Select OK when you have finished all of the changes.

BEAMS

Bevel Flanges:
Enter values for “Bevel Flanges for field welds”.

Shop bolted Clips and Stiff:

#1 & #2 are the gages for shop bolted clips. #3 is the setback from the heel to end of web. #4 is the clearance between beams at the flange block. #5 is the clip thickness. Enter 0 for item #6 to make the length of stiffeners and wing plates that welded to both flanges equal to beam depth minus 2 times flange thickness. Otherwise the length will be shortened by the amount you enter.



Blocks

Style:
This applies to level beams only. Choose the style you want by picking the image. If you are saving CNC data, burn data for blocks will be saved with the block length dimension.

Depth:
Here you select how the block depth is calculated. Select either to use the “k” dimension of the support or flange thickness, “tf”, plus a clearance value. If you use the second option and put a check mark in “Min “k” of blocked beam” the depth will be the greater of tf plus clearance and the “k” dimension of the blocked beam.

Dimension blocks to:
This controls where blocks, copes and cuts are referenced from at the ends. The location is for the horizontal dimension. If you are saving CNC, burn data for blocks will be saved with the horizontal dimension. The burn length will be the same as the horizontal dimension of the block.

Wing PL block option:
Extending beam block will cause wing plates to be cut square and the length of the block on the beam will be increased to clear the plate. Notching the wing plate will cause the wing plate to be cut to clear the beam flanges.

Horizontal dimension are: Work length is at: Overall length is at:
These three settings establish the location of control dimensions. The setting of the location of horizontal dimensions has no effect if the beam is slopping. In that case, the location is

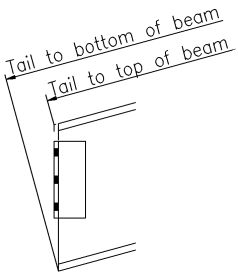
determined by the direction of the slope. Horizontal dimensions will be above beams slopping down to the right and below beams slopping up to the right.

Conn for end of beam w/holes in web:
Clearance for non-standard shear bar:
This clearance will be used if you do not have a standard shear bar configuration for the bolt size in use. The clearance can be (1) from centerline of supporting web or the face of a column flange or (2) from the face of support. If a standard shear bar configuration can be used, the end clearance is the gage of the shear bar minus the end of beam edge distance.
Shear bar configurations you can have standard marks for bars with round holes or bars with short slots. Put a check in the box "Use standard shear bar with slots" to use the mark for the bar with short slots.

Extension and tails at left end:
This controls where tail or running dimensions and the extension line for the overall length are referenced from at the left end. You can have both to beam, both to clip or extension to clip and tails to beam.

Minus dimensions framing into beams:
This determines if you want to give an extra 1/16" clearance at the end of a beam with clips when framing into a beam web.

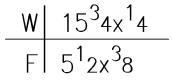
Beams sloping up to the right:
For this condition you select whether tail dimensions come from the left end of the beam along the top flange or the bottom flange. If you are saving CNC the default is bottom flange.



Use max rows in end connection:
Check this if you want to use maximum rows in all beam end connections.

Dimension clip edge distance:
Check this if you want to put a dimension to the bottom edge of end connection angles.

Show beam properties:



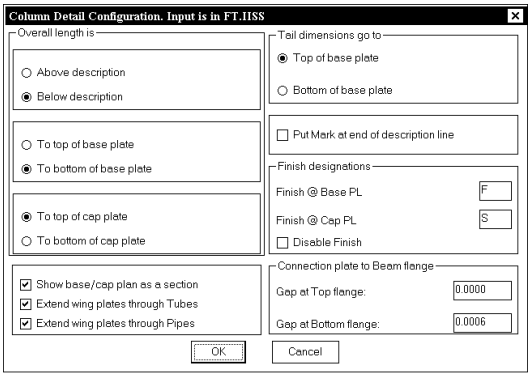
Check this if you want to put this symbol showing the depth and web thickness, (W), and the flange width and thickness, (F), under the main mark.

Show support member size:
Check this if you want to put the support member size at the end of the beam detail.

Show loads on end of beam:
Check this if you want to put the loads at the end of the beam detail on a layer called LOADS.

Grade of steel:
Select the grade of steel that you want as the default for Angles, Channels, Flange shapes, Plates/Bars, Pipes and Tubes. Enter the grade for studs.

COLUMNS



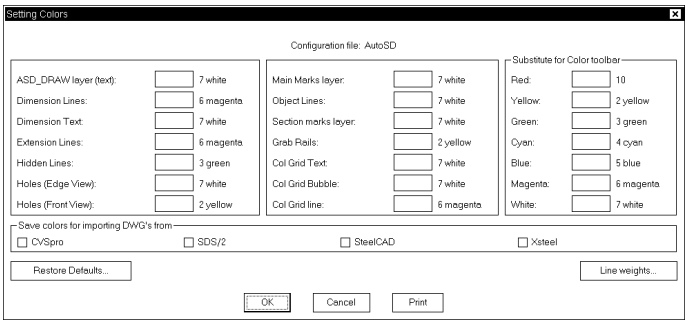
This box is for setting the parameters for dimensioning columns. It controls where the overall length and tails are dimensioned to, the location of the overall length, the symbol to be used for finishing each end of the column and whether to put the mark at the end of the description line or not. "Connection plate to Beam flange" controls the gap between the beam flanges and the horizontal plates that will be welded to the column when you save the beam to column file while detailing the beam. If you put a check in "Extend wing plates through Tubes" or "Extend wing plates through Pipes", all non-skewed wing plates will be detailed going through the tube or pipe column.

EDITING A COLOR CONFIGURATION

Pulldown: AutoSD > Customer Configuration
Ribbon: AutoSD I > Shop bill

Pick Color list, select one of the names under "File name" and pick Edit.

The first time you open the configuration box and pick "Colors list" you will have 5 configuration files in it. You can edit and use these files. You can rename them or copy them and make more files. If you have more than one file you can delete them but you cannot delete the last one. There will always be at least one file. All of the color configurations are stored in one file called custCLR.cfg located in your autosd folder. You can also import individual configurations from other custCLR.cfg files.



Select a color configuration file and pick the EDIT button. The Setting Colors Configuration box pops up.
These files store the colors for dimension lines, extension lines, dimension text, section labels, text, holes, hidden lines, object lines, Main marks, Grab Rails Column Grid text, Column Grid bubble, Column Grid line, Color toolbar and Line toolbar. The color of layer ASD_DRAW is used to control the color of text, the layer ASD_SECT is used for the color of section labels and the layer ASD_MARK is used for the color of main marks.

Substitute for Color toolbar: The color used for the Color and Line toolbars are controlled by these settings. The actual color of the icons in the toolbars will not change but you can vary the color used by each icon. The color used by the Red icon, for example, can be changed to any shade of red or any other color.

To change a color, select a color swath to display the color choices. Pick a color and pick OK.

The default color configuration files copied to your AUTOSD folder when AutoSD is installed are AutoSD, CVSpro, SDS/2, SteelCAD and XSteel. These are just arbitrary file names and have no association with the color configuration when importing DXF files into AutoSD for CVSpro, SDS/2 or XSteel. You can rename these files as you like.

Save colors for importing DWG's from:

Below the color choices is a row of check boxes. When you import a DXF file using the command "DXF into AutoSD" the colors of some of the objects will be changed to match the colors in the configuration. You can change the colors in these configuration files by putting a check in the box for CVSpro, SDS/2, SteelCAD or XSteel.

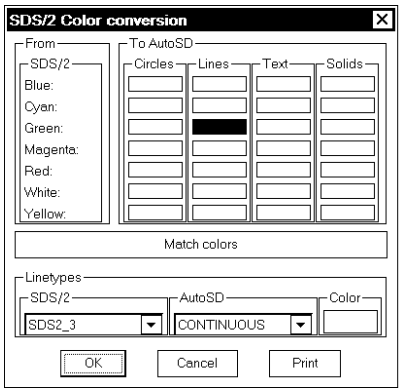
When you pick OK, the colors will be saved to the file custCLR.cfg for the file name selected in the Color list box and to any and all names that are checked.

The colors used for importing DXF files are saved in the following files.

File name	Configuration use
CFG-SDS2.LSP	SDS/2
CFG-CVS.LSP	CVSpro
CFG-SCAD	SteelCAD
CFG-XSTL.LSP	Xsteel

Exception:

The file CFG-SDS2.LSP for SDS/2 is NOT used for converting colors when importing dxf files from SDS/2. It is only used to set the colors for detailing when you run [AutoSD > Format or AutoSD I > Utilities] > Setup and select the SDS/2 option.

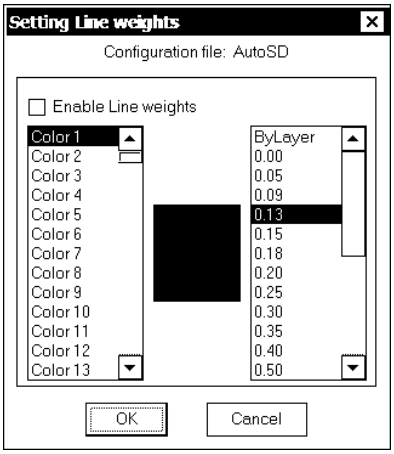


Converting colors when importing dxf files from SDS/2 is handled by the command [AutoSD > Tools or AutoSD I > Utilities] > DXF into AutoSD > Configure SDS/2 colors. The dialog box shown to the left is used to configure the colors. The list under SDS/2 is the color of the object in the dxf file. The colors under "To AutoSD" is what each color will be changed to for each type of object. Solids are for arrowheads.

Names for specific line types under "Line types" will be changed to the name in the list next to each name. The color here will override the colors shown above Line types.

After making changes, pick OK to save those changes. Picking Cancel will remove all changes made to colors and line weights.

LINE WEIGHTS



At the bottom of the customer colors configuration dialog box is a button that opens a line weight dialog box shown to the right. This option is for setting line weights for plotting lines of different thicknesses by line weight. Any settings here will override line weights based on color when plotting.

Putting a check in the box for "Enable Line weights" will cause the program to use the values set here for line weights. Un-checking the box will disable this feature.

After making changes, pick OK to save those changes. Picking Cancel will remove all changes.

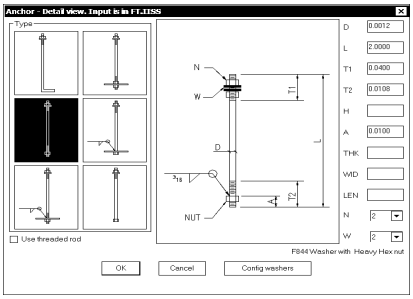
ARROWHEADS

Arrowheads in previous versions of AutoSD used a block called Dimarrow. This has been changed. You will need to open the Dimension Style dialog box, pick the Lines and Arrows tab and set the 1st, 2nd and Leader arrowhead to Closed filled. You may select any of the arrowhead types from the list but Closed filled will look the same as the old Dimarrow block. If you do not change this setting your arrowheads my not be the correct color.

Chapter 2 - Erection Tools

ANCHOR BOLT SIDE VIEW

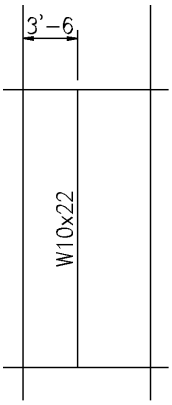
Pulldown: AutoSD > Erection Tools > Anchor Bolt Side View
Toolbar: Erection Tools
Ribbon: AutoSD I > Erection Tools



This program will draw the side view of any of the six different anchor bolts. The insertion point is the underside of the washer and the bolt can be rotated to any angle. The scale is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.

BEAM LINES w/DESC

Pulldown: AutoSD > Erection Tools > Beam Lines w/Desc
Toolbar: Erection Tools
Ribbon: AutoSD I > Erection Tools

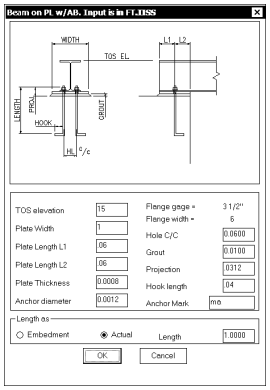


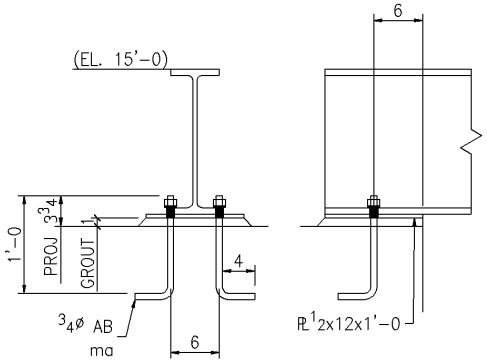
This program draws a line with several options. You can put a moment symbol, a solid filed triangle, at the ends. You can include the camber with a custom prefix. You can include shear studs with a custom bracket character on each side of the number. The stud count can be on the same side of the line as the member size or the opposite side. The member size is always printed, the camber and stud count are optional.

Pick 1st end of beam or [Select line] <Select line>:
Pick a point to draw just one beam from point to point or press Enter or "S" and enter to draw multiple beams between existing lines. The member size will be placed at the center of the line. Dimensions from beam to beam can be placed automatically for the multiple beam option. Three existing lines are required to start if you are drawing multiple beams. Select a line at each end of the beam for a start and stop point. They do not have to be parallel. Select a third line parallel to the beam to be drawn as a reference for the first beam and select which side of this reference line to draw the beams. The program will continue to ask for the distance to the next beam, the offset at the 1st end and the offset at the 2nd end. A positive offset is for a cantilevered condition. Enter the dimensions and select the member size of the next beam. When asked for the distance to the next beam you can enter 0, (zero) to end the command or press Ctrl C or Esc to stop the command. All beam lines will be placed on a layer called BEAM. The scale is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.

BEAM ON PLATE w/AB

Pulldown: AutoSD > Erection Tools > Beam on Plate w/AB
Toolbar: Erection Tools
Ribbon: AutoSD I > Erection Tools





This program will detail the end view and web view of a beam sitting on a bearing plate with anchor bolts showing the projection, grout, bolt length, hook length, bolt diameter, hole pattern and plate size. This detail is used on anchor bolt setting plans. You can use any of the six anchor types shown in "Anchor Bolt Side View" on page 27 plus an expansion anchor and an epoxy anchor. The scale is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.

BEARING PLATE EDGE VIEW

Pulldown: AutoSD > Erection Tools > Bearing Plate Edge View
Toolbar: Erection Tools
Ribbon: AutoSD I > Erection Tools

This program draws the sectional end view of a typical bearing plate used for joists or beams when bearing on a masonry wall. You enter the plate width and stud length. You can have one or two studs. No dimensions will be shown. The scale is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.

BRACE LINES w/DESC

Pulldown: AutoSD > Erection Tools > Brace Lines w/Desc
Toolbar: Erection Tools
Ribbon: AutoSD I > Erection Tools

This program will draw a line to represent a brace and place the member size at the center of the line and an option to place a solid filed triangle moment symbol at the ends. Select a member size and enter an offset at each end. A negative offset will pull the brace back from the selected point. Select a point at each end of the brace for a start and stop point. All information will be placed on a layer called BRACE. The scale is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing" command.

BLOCK WALL SECTION

Pulldown: AutoSD > Erection Tools > Block Wall Section
Toolbar: Erection Tools

Ribbon: AutoSD I > Erection Tools

This program draws a cross section of a 4", 6", 8" or 12", (100mm, 150mm, 200m and 310mm), block wall. The first point you pick will be a corner point of the first whole block on one face. The second point may be above or below the first point. The third point picked determines if the face location picked was the right or left side. The blocks are drawn 7 5/8", (193.7mm), tall with a 3/8", (9.5mm), mortar joint. The scale is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing" command.

BRICK w/BLOCK WALL SECTION

Pulldown: AutoSD > Erection Tools > Brick w/Block Wall Section
Toolbar: Erection Tools
Ribbon: AutoSD I > Erection Tools

This program is similar to Block Wall Section above but combines brick and block into one program to draw a cross section of the wall. You input the total thickness of the wall. The scale is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing" command.

BRICK WALL SECTION

Pulldown: AutoSD > Erection Tools > Brick Wall Section
Toolbar: Erection Tools
Ribbon: AutoSD I > Erection Tools

This program draws the cross section of a brick wall. The first point you pick will be a corner point of the first whole brick. The second point may be above or below the first point. The third point picked determines if the face location picked was the right or left side. The brick are drawn 3 3/4", (95.25mm), wide with a 3/8", (9.5mm), mortar joint. 3 brick equals one block in height. The scale is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.

BLOCK WALL PLAN VIEW

Pulldown: AutoSD > Erection Tools > Block Wall Plan View
Toolbar: Erection Tools
Ribbon: AutoSD I > Erection Tools

This program draws the plan view of a 4", 6", 8" or 12", (100mm, 150mm, 200m and 310mm block), block wall. The first point you pick will be a corner point of the first whole block at one end of the wall. The second point will be the corner point at the other end of the block wall. The third point determines which side of the line between the first two points the blocks are on. The blocks are drawn 15 5/8" long with a 3/8" mortar joint. The scale is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.

BRICK w/BLOCK WALL PLAN VIEW

Pulldown: AutoSD > Erection Tools > Brick w/Block Wall Plan View
Toolbar: Erection Tools
Ribbon: AutoSD I > Erection Tools

This program is similar to Block Wall Plan View above but combines brick and block wall plan view into one program. You input the total thickness of the wall. The scale is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.

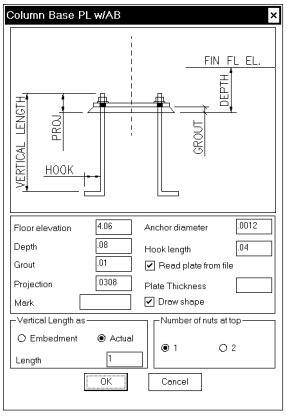
BRICK WALL PLAN VIEW

Pulldown: AutoSD > Erection Tools > Brick Wall Plan View
Toolbar: Erection Tools
Ribbon: AutoSD I > Erection Tools

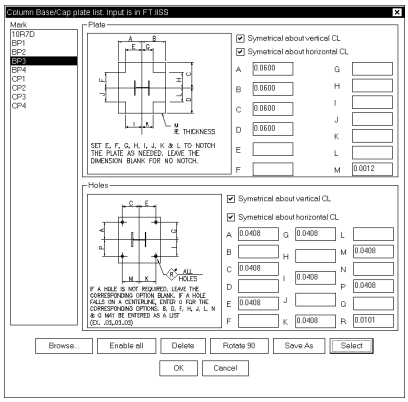
This program draws the plan view of a brick wall. The first point you pick will be a corner point of the first whole brick at one end of the wall. The second point will be the other end of the brick wall. The third point determines which side of the line between the first two points the bricks are on. The bricks are drawn 3 3/4" wide, 7 5/8” long and with a 3/8" mortar joint. 2 brick equals one block in length. The scale is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.

COL BASE PLATE w/AB

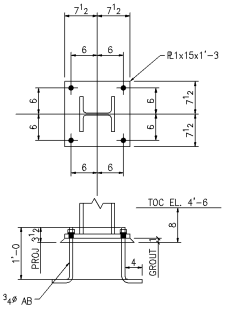
Pulldown: AutoSD > Erection Tools > Col Base Plate w/AB
Toolbar: Erection Tools
Ribbon: AutoSD I > Erection Tools



This program will detail the plan view and section view of a column base plate with anchor bolts showing the projection, grout, bolt length, hook length, bolt diameter, hole pattern and plate size. This detail is used on anchor bolt setting plans. You can use any of the six anchor types shown in “Anchor Bolt Side View” on page 27 plus an expansion anchor and an epoxy anchor.

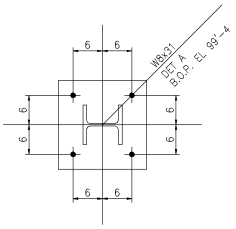


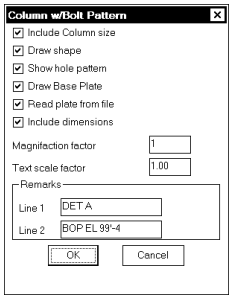
The dimensions for the plate are entered in the dialog box to the left where they can be read from the file for other details. The scale is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.



COLUMN w/BOLT PATTERN

Pulldown: AutoSD > Erection Tools > Column w/Bolt Pattern
Toolbar: Erection Tools
Ribbon: AutoSD I > Erection Tools

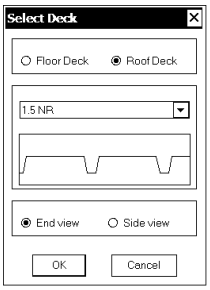




This program will draw the end view of a column to scale and place a line at 45 degrees in the quadrant you select and place the member size on the top side of the line and up to two reference notes on the bottom side of the line. The size of this text is controlled by the "Text scale factor". All of this is on a layer called "Column". It will draw and dimension a bolt pattern with or without base plate and put this on a layer called "Anchors". You have the option of drawing just the column, just the description line or just the bolt pattern. The scale is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command and can be scaled up by entering a magnification factor greater than 1.

DECK EDGE VIEW

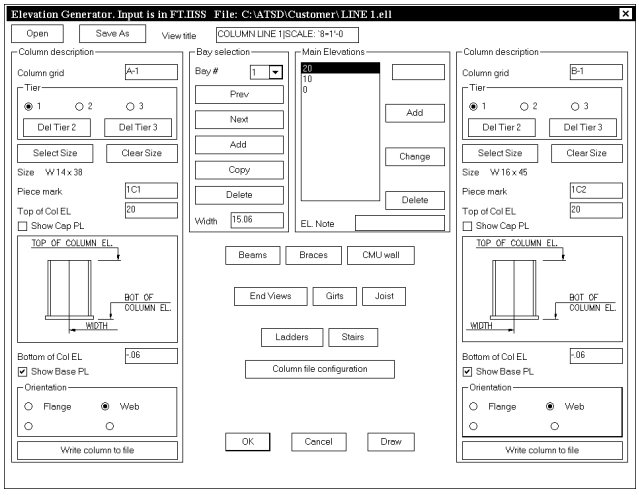
Pulldown: AutoSD > Erection Tools > Deck Edge View
Toolbar: Erection Tools
Ribbon: AutoSD I > Erection Tools



This program will draw the edge view of floor and roof deck. The scale is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command. Deck descriptions are stored in the data file US_deck.dat in your autosd\data folder. You may edit this file to add more descriptions to it using the database editor. See chapter 4 and Appendix B.

ELEVATION GENERATOR

Pulldown: AutoSD > Erection Tools > Elevation Generator
Toolbar: Erection Tools
Ribbon: AutoSD I > Erection Tools



This program is for creating elevation views for erection drawings and plan views of stairs. The information is entered into dialog boxes and saved to a file name and location of your choosing. One file holds one view. You can create elevation views

showing beams, braces, CMU walls, columns, end view of shape on column, girts, joist, ladders and stairs. The views can be drawn to various scales independent of the scale that your drawing is set up to.

Input is entered in bays. One bay is the area between two adjacent columns. Bays are created from left to right.

Main dialog box

Open: Lets you select existing elevation files to open for editing.
Save As: Lets you save a file that has not yet been saved or lets you save an existing file to a new file name. An elevation cannot be drawn until the file is saved.

View title: This is name of the elevation view and will be printed below the elevation detail. If you want additional text lines printed under the title, separate the text lines with the "|" character.

Example:

COLUMN LINE A | SCALE: `4" = 1'-0" Will be printed as: **COLUMN LINE A**
SCALE: $^{1}4$ = 1'-0

Note that the inch marks will be removed from the printed text.

The main dialog box is divided into 4 sections. Left side, Right side, Top center and Bottom center.

Left side and Right side sections

These are for entering the column information of the current bay. The first edit box is for the optional column grid. If no grid designation is needed just leave this box blank. You can have up to three tiers. The "Piece mark" edit box is for the shipping mark of the column of the currently selected tier. If no piece mark is needed just leave this box blank. The "Top of Col EL" is the elevation at the top of the column or the column splice for multi-tier columns. Picking "Del Tier 2" will delete the column for tier 2 and tier 3 and puts you back to having only one tier. Picking "Del Tier 3" will delete the column for tier 3 and puts you back to having two tiers. Picking "Select Size" lets you pick a column size for the currently selected tier. Picking "Clear Size" will remove the column size for the currently selected tier. The "Bottom of Col EL" is the elevation at the bottom of the column or the column splice for multi-tier columns. "Orientation" lets you select how the column is viewed. "Show Cap PL" and "Show Base PL" toggles cap and base plates on if selected and off if cleared.

Columns are optional. If a column is not required you must leave the column size blank. If you have an elevation detail with no columns such as a stair elevation inside block walls you need to enter a "Top of Col" elevation for one of the columns that is the height of the block walls or higher and a "Bottom of Col EL" that is the elevation of the bottom of the lowest stair. These elevations determine where the bay width dimension is located.

Top center section

This is for adding or changing Bay information and entering the Main elevations.
Bay selection: Bays are numbered in sequence from left to right starting with #1. Picking "Add" will add a bay. The "Bay #" will increase by one and you will be place in the last bay at the right end of the elevation. You can add a bay at any time. Picking "Copy" will add a copy of the last bay to the right end of the elevation. "Copy" will only be available when you are viewing the last bay. Picking "Delete" will delete the current bay. To select a different bay you can pick the "Next" or "Prev" buttons or you can pick the down arrow to the right of "Bay #". This will drop down the list of all bays where you can select the number of the bay you want. All bays must have a width. Enter the width from center line of the column on the left to the center line of the column on the right in this edit box.
Main Elevations: Elevations entered here will be dimension on the left side of the elevation detail and apply to all bays. To enter an elevation type a number in the edit box and pick "Add". Repeat this until all elevations have been entered. To change an elevation pick the elevation, type in a new number in the edit box and pick "Change". Picking "Delete" will remove the currently selected elevation.
EL. Note: If you want to add a note such as "First Floor" or "2nd Floor" to the detail pick the elevation you want the note associated with and then enter the note in this edit box.

Bottom center section

This is for entering information in the current bay by picking a button that opens a new dialog box for that member.

The following instructions are similar for Beams, Braces, CMU wall, End Views, Girts, Joist, Ladders and Stairs.

The top of the dialog box will indicate which bay you are in by listing the column grids and the bay number. The default will be the current bay in the main dialog box. You can change bays by picking the "Next Bay" and "Prev Bay" buttons. Pick "Add" to add a member. A dialog box will pop up for you to select a member size and elevation. The elevations listed under "Main Elevations" will be in this dialog box for selecting or you can enter a different elevation in the edit box. If the member is sloping, simply put a different elevation at each end. The "Change" button lets you change the size and/or elevation of the currently selected member. The "Delete" button will remove the currently selected member. The size and elevations will be displayed in the list box.

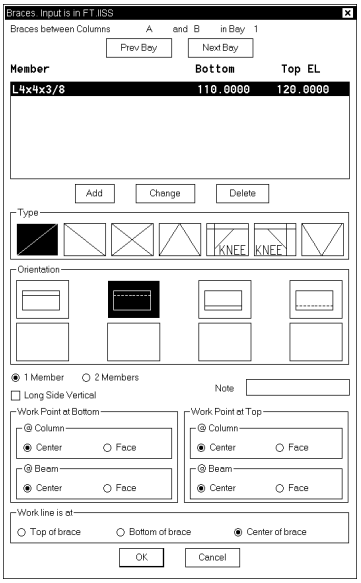
Beams

If you have a beam attached to only one column you need to put a check in "Stub Beam" box and select either "On Left Col" or "On Right Col" of the current bay. Next, select which side of the column the beam is on and enter the "Length of beam" in the edit box. This is the distance from the center line of the column to the end of the beam. If you have a beam that extends over the top of a column you need to put a check in the "Cantilever" box. Next, select which column the beam cantilevers and enter the "Length of beam" in the edit box. This is the distance from the center line of the column to the end of the beam.

You can add a note to the left and/or right ends of a beam. This can be an end reaction, piece mark or other note. Separate the text lines with the "|" character to put multiple lines of text at one end.
Example: **23k | 48B3** will be printed as: **23k**
48B3
with the "23K" above the beam line and all other text below the beam line.

Below this is the "Orientation". The selections available will depend on the shape of the beam. "W" shapes will either be "Web" or "Flange" view. Channels can be toed Up, Down, NS or FS. The image above and at the right end of Orientation will show an icon of what has been selected.

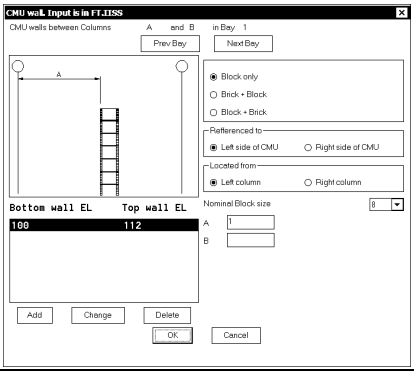
Braces



Pick Add to add a brace to the current bay. The top and bottom elevation for the brace are normally the top elevation of the beam it frames to.
Select the "Type" of brace by picking an icon image. Next select the "Orientation". The icons available will depend on the shape of the brace. If the brace is an angle select whether it is "1 Member" or "2 Members" and if you want a designation for the "Long Side Vertical" added to the detail put a check in that box. If you need to add additional text under the center of the brace put it in the "Note" edit box. Separate the text lines with the "|" character to put multiple lines of text.
Example: **(47k %%p) | 21D1** will be printed as: **(47k +-)**
21D1
Work Point at Bottom: (of brace) and **Work Point at Top:** (of brace)
Select where the work line of the brace will intersect the beam and column, either "Center" or "Face".
Work Line is at: Select where the work line of the brace is located.

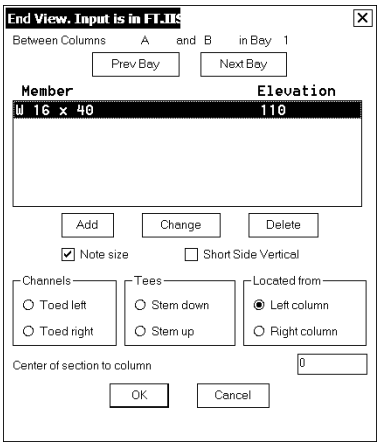
After making all selections for one brace pick "Add" to add another brace. The selections of the currently selected brace will be the defaults for the new brace. Pick OK to save and close the Brace dialog box. Pick Cancel to close the Brace dialog box without saving the changes.

CMU wall



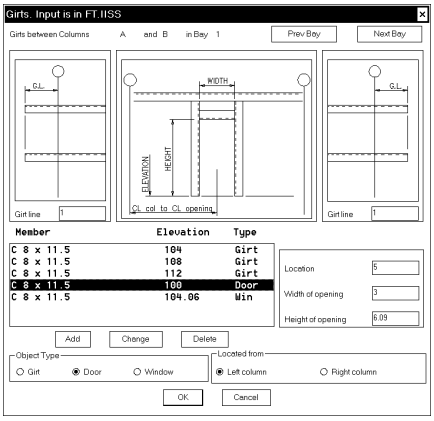
The wall can include brick on either side. Dimension A can be referenced from either column line and from either side of the wall. Block widths available are 4, 6, 8, 10 and 12.
The dimension locating the wall from the column will be drawn above the top of the wall. If you have two walls the same height located from the same column the dimensions will overwrite. To avoid this locate the left wall from the left column and the right wall from the right column.

End Views



Putting a check in the box for "Note size" will add a leader with the member size to the detail. "Short Side Vertical" is for the orientation of tubes. Angles and channels can be toed left or right. Angles and Tees will have the leg or stem pointed up or down. The dimension locating the end view can be from the left or right column.

Girts



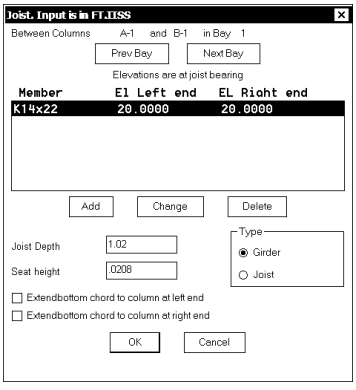
The left and right side of the dialog box show an image for a "Girt line". If the girts extend past the column line at the left end enter a distance for the "Girt line" under the left side image. A negative distance will stop the girts short of the column line. The girt line information will be available no matter which bay you are editing but will apply only to the first and last bay of girts.

After adding a girt, select the "Object Type". It can be a Girt, Door or Window. The center image is changed to show which type is selected and the type is changed in the display list. If a door or window is selected the "Location", Width of opening" and "Height of opening" are enabled to the left of the display list and values must be entered for each. The location can be from the left column or from the right column as selected under "Located from".

After making all selections for one girt pick "Add" to add another girt. The selections of the currently selected girt will be the defaults for the new girt. Pick OK to save and close the Girt dialog box. Pick Cancel to close the Girt dialog box without saving the changes.

Joist

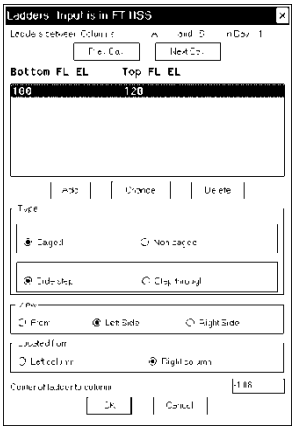
The elevations entered for each end are to the joist bearing.



The joist size is taken as a literal string for labeling. The Depth and seat height is entered in the joist dialog box for each joist. Remember to use the correct format for dimensions as shown at the top of the dialog box.

After making all selections for one joist pick "Add" to add another joist. The settings of the currently selected joist will be the default for the new joist. Pick OK to save and close the Joist dialog box. Pick Cancel to close the Joist dialog box without saving the changes.

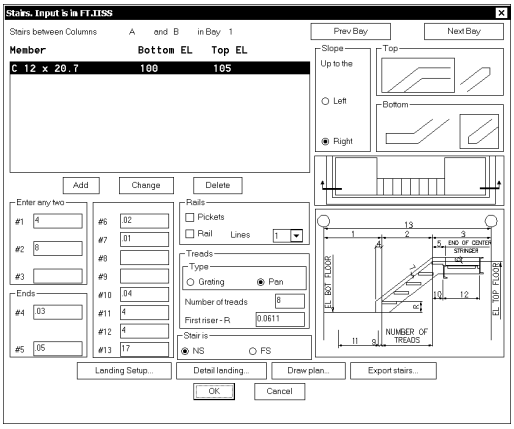
Ladders



Under "Type" select "Caged" or "Non-caged" and "Side step" or "Step through". Under "View" select "Front" "Left Side" or "Right Side". The location can be from the left column or from the right column as selected under "Located from". The dimension for the location is entered in the edit box for "Center of ladder from left column"

After making all selections for one ladder pick "Add" to add another ladder. The selections of the currently selected ladder will be the defaults for the new ladder. Pick OK to save and close the Ladder dialog box. Pick Cancel to close the Ladder dialog box without saving the changes.

Stairs

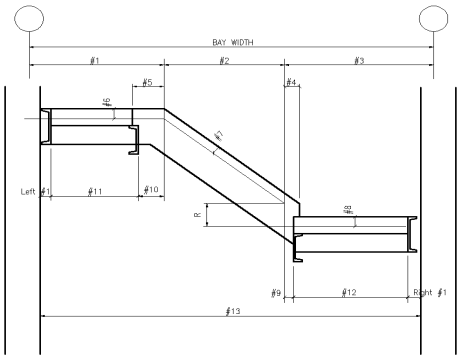


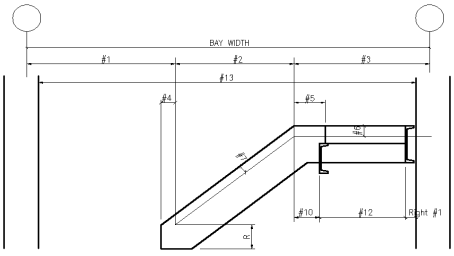
To the right of the display list are selections for "Slope", "Top" and "Bottom". The slope orientation of the stair can be up to the left or up to the right. Changing it will change the images shown. Select the top and bottom condition and the image below will change to reflect your selections.

Enter any Two: For dimensions #1, #2 and #3 you only need to enter 2 of these. Those two are subtracted from the bay width to get the third.

Ends: Dimension #4 is from the work point at the bottom end to the end of the stringer and #5 is from the work point at the top end to the end of the stringer.

Elevation view of stair image:





#6 is from top of upper floor to top of frame member.

#7 is from nosing line to top of stringer.

#8 is from top of lower floor to top of frame member.

#9 is from the work point at the bottom tread to the back of the header on the lower landing. This dimension is required even if this is the ground floor. It can be 0.

#10 is from the work point at the upper floor to the back of the header beam.

#11 is the depth of the landing on the left side from back of header to back of wall member.

#12 is the depth of the landing on the right side from back of header to back of wall member. #11 & #12 locate the face of the CMU wall shown in the landing setup.

#13 is a global dimension used for all stairs in one bay. The dimension can be the same as the bay width or inside distance of CMU walls at each end of stair. If the stair frames into a floor beam on the column center line on the left, for example,

dimension #11 will not be used but must still be entered. You must select "None" for the frame type in the Landing Setup in this case.

Rails: Select either Pickets or Rail or neither. If Rail is selected you need to select the number of lines.

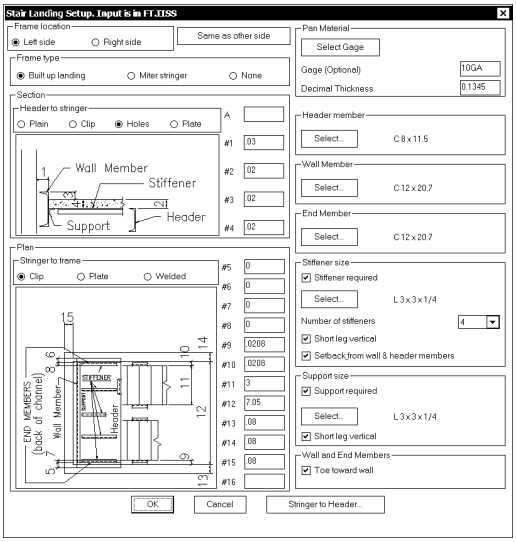
Treads: Select the type of Grating or Pan. The image to the right will change to reflect your selections.

Enter the "Number of treads" in the edit box and press the enter key or tab key and the first riser will be calculated and put in the "First riser - R" edit box as the default. You can change it or leave it as-is.

Stair is: Either NS or FS. The image of the plan and elevation view will change to reflect your selection.

After making all selections and entering all values for one stair, pick "Add" to add another stair. The selections of the currently selected stair will be the defaults for the new stair. Pick OK to save and close the Stairs dialog box. Pick Cancel to close the Stairs dialog box without saving the changes.

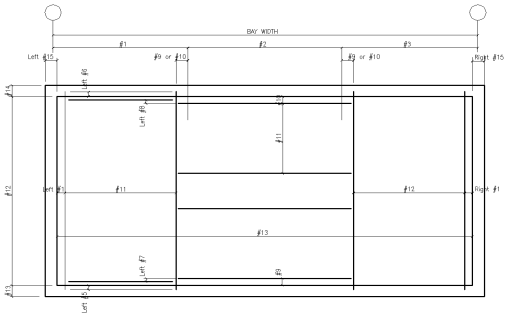
Landing Setup



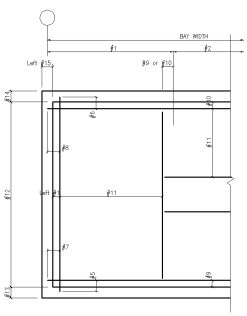
Pick this button to configure the landings for both right and left side. There is only one landing configuration per elevation view. The same configuration is used on all floors in one elevation view. The configuration set here will be the default for the next elevation file.

Frame location: Left side, Right side: Select the side that you want to configure. When you have finished that side select the other side. If you want this side configured to be the same as the first side pick the "Same as other side" button.

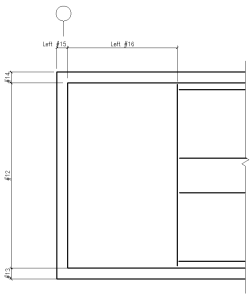
Frame type: The frame type can be built up or mitered, ("dog leg" - shop welded to stringer) or none for framing the stringer to a floor beam. The one you select is reflected in the Plan input at the bottom of the dialog box. Fill out all information for both right and left sides.



BUILT–UP LANDING



MITERED LANDING



LANDING – NONE

Section: This is enabled if you select either built up or mitered frame type. This shows a section through the frame showing the end view of the header beam with a CMU wall. In a built up frame the header is one side of the frame so the header to stringer connection options are disabled. #1 is from face of CMU to back of wall member. #2 is from top of floor to top of header. #3 is from top of wall member to top of floor.

#4 is floor thickness. “A” is from back of header to center of holes if the frame type is mitered and the header to stringer connection is holes.

Plan: Dimensions shown in plan for the landing. #5 & #6 are from back of frame end member to end of wall and header members. #7 & #8 are from back of built up frame end member to back of stringer or from back of wall member to ends of mitered stringer. Only one end at each corner should extend past for a mitered frame, so, one dimension at each corner should be 0. #9 & #10 are from back of stringers to edge of floor opening or face of CMU walls. #11 is width of stairs. #12 is from edge to edge of floor opening or face to face of CMU walls. #13 & $14 are the thickness of the CMU walls. These can be 0 for no walls. #15 is from the column center line to the inside face of the CMU wall. The outside line of the CMU end wall is the center of the column line. Enter 0 for #15 for no CMU wall.

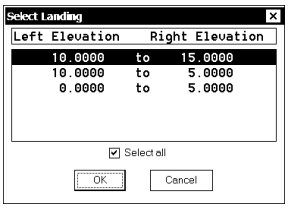
The right side of the dialog box is for entering the landing member sizes and selecting which way the wall members are toed.
Stringer to Header: Enter dimension #1, #2 & #3 to locate the holes for this connection.

Detail landing

If landings are built-up as a frame, and you have the Stairs and Rails package, they may be detailed by picking this button. It will close the Elevation generator program and open the Landing Frame program. The input for the landing frame will be exported for the elevation selected and imported into the Landing Frame program.

Draw plan

Pick this button to draw a plan view of each landing. A list will be displayed, as shown to the right, for you to select an individual plan elevation or put a check in the box “Select all” to draw all plans.



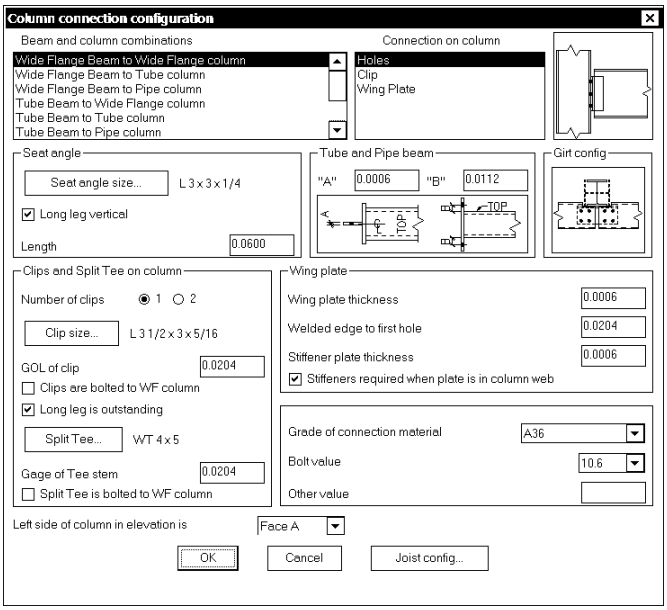
Export stairs

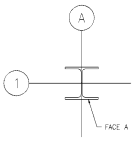
If you have the Stairs and Rails package, pick this button to export all stringers in the current bay. They can be detailed by the Stringer program. The file name for each stringer will begin with the file name of the current open Elevation Generator file plus the bay number plus the bottom and top elevations of the stringer.

Column file configuration

If you have the Beam and Column package you can create column files from the elevation views. Pick the Bay that you want and then pick the “Write to column file” button under the right side or left side column. If the column is spliced you will need to select the tier as well and make a separate file for each tier. After selecting an existing column file to append to or replace or creating a new column file, a configuration dialog box pops up. You can also open this box by picking the “Column file configuration” button on the main dialog box. This box pops up every time for you to select the proper column face but it needs to be configured only

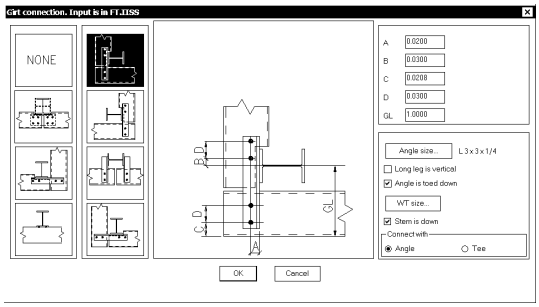
once or when the need requires it. If you have girts you will need to select the correct connection for the current column by picking the Girt configuration image button.





If you have two elevation views, you can create a single column file that contains the connections for members in both elevations. See the picture to the left that represents the plan view of a column at lines "1" and "A". If you are looking at line "1" the face on the left side of the column in the elevation view would be "D". Looking at line "A" the face on the left side of the column in the elevation view would be "A". After creating the column file for the first elevation view, append the results of the second view to the existing file. The two views may be processed in either order.

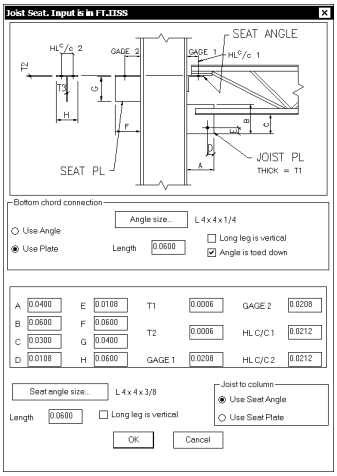
Girt configuration



Pick the Girt config image button and select the proper girt to column connection from the image icons on the left side. Enter the dimensions for the connection and select the angle and WT size.

Pick OK to save the settings. Picking cancel will cause you to lose all changes made to the configuration.

Joist configuration



Select the Joist config button and select the seat type and enter values for all options available. Pick OK to save the settings. Picking cancel will cause you to lose all changes made to the configuration.

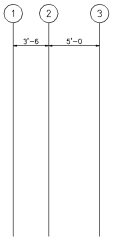
Draw

This will save all changes to the file and open up the "Select Scale" dialog box. Pick the scale you want the elevation drawn to and pick OK. You will be prompted for the insertion point while a box is drawn around the cross hair location. The box is the extents

of the elevation view and will move with the cross hairs. After picking a point the box disappears and the elevation is drawn.

GRID LINES w/DIM

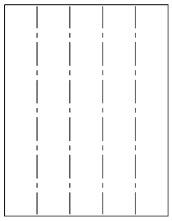
Pulldown: AutoSD > Erection Tools > Grid Lines w/Dims
Toolbar: Erection Tools
Ribbon: AutoSD I > Erection Tools



This program will draw grid lines with column grid bubbles and optional dimensions between grid lines. You draw the first grid line. The end you start with is the end the grid bubble will be on. After the first grid line and bubble has been placed you select one side of the line for the placement of other grid lines and bubbles and you select the location of dimensions if any are required. After that all you input is the distance to the next line and the designation for the grid bubble. Press ESC or enter 0 for the distance to the next grid line to end the program. Entering the Grid mark as "150|11" will stack the text with "150" on top and "11" underneath. The scale is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.

JOIST LINES

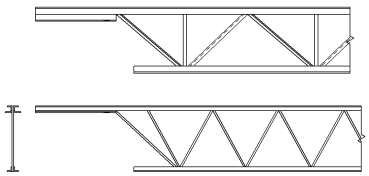
Pulldown: AutoSD > Erection Tools > Joist Lines
Toolbar: Erection Tools
Ribbon: AutoSD I > Erection Tools



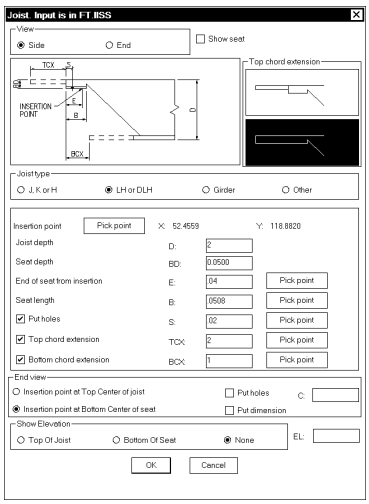
This program is similar to the beam lines program. This program will place joist inside a four sided area. The boundaries of this area do not have to be square or parallel. The joist lines will be put on a layer called JOIST. At the prompt "(FT.IISS) <Joist spacing>/Number of equal spaces:" you can enter a real number for the spacing or the letter N, (upper or lower case), to enter the Number of equal spaces. The spacing is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.

JOIST SIDE or END VIEW

Pulldown: AutoSD > Erection Tools > Joist Side or End View
Toolbar: Erection Tools
Ribbon: AutoSD I > Erection Tools



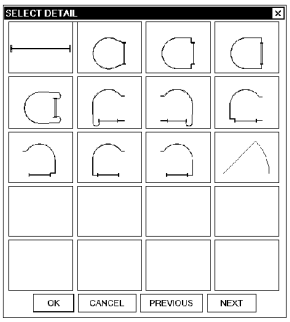
This program draws the side or end view of open web joist and joist girders. Joist sizes are (H, J or K), (LH or DLH), Girder and Other. If "Other" is selected you will be prompted to enter the size of the top chord, bottom chord and web member.



Side View: You must pick the insertion point for the bottom of the seat at the edge of support and a point for the length of the joist before you can enter all of the variables. For the location of the end of seat from the insertion point, seat length, location of holes, top chord extension and bottom chord extension you can enter a length or pick a point. There are two types of top chord extensions. The one selected will be shown with a black background.
End View: Put a check in "Show seat" if you want the seat drawn. Insertion point can be at the top of the joist or bottom of the seat. You can select to put holes in the seat with a dimension to them.

A reference elevation can be put on both side view and end view.
The scale is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.

LADDERS

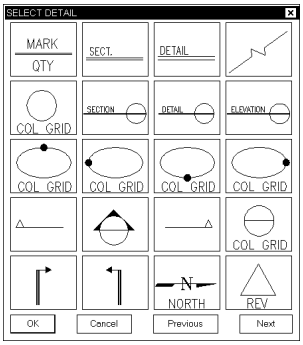


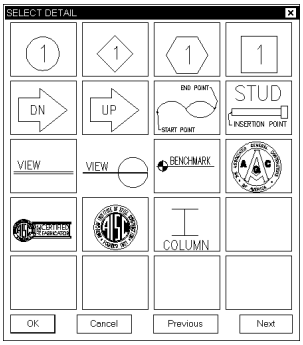
Pulldown: AutoSD > Erection Tools > Ladders/Gate
Toolbar: Erection Tools
Ribbon: AutoSD I > Erection Tools

This will display an icon menu for the selection of plan views of ladders with and without cages for use in creating erection drawings. The insertion point is the middle of the rung. The scale is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.

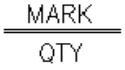
MARKS / SECT / TAGS

Pulldown: AutoSD > Erection Tools > Marks / Sect / Tags
Toolbar: Miscellaneous
Ribbon: AutoSD I > Miscellaneous



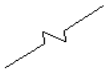


This will display an icon menu for the selection of piece mark, section marks, break lines, north arrow, column bubble, column, AISC symbol, revision symbols and UP and DOWN arrows.



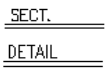
This places attributes in the drawing for piece marking an item and giving its quantity. There are two formats: MARK over QUANTITY and QUANTITY - BEAM - MARK. If

you are saving CNC data the mark will also have two hidden attributes for the size and length. If you change the size of the member or the length in the detail view you will need to change the values of these attributes with the ATTEDIT command. The length will be in the FT.IISS format. There is one more attribute that is left blank. When you edit the attributes the prompt for this attribute is REF: and is used for inserting a single line reference note under the mark.

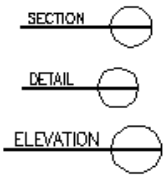


This is for drawing "break" lines. Pick the start and end points and the "break" line will be drawn. The line will extend 1 1/2", at a scale of 1" = 1', past each point picked.

These two are for cutting sections. A letter or number of your choice is placed with the symbol.



These two are for labeling a section or a detail. When either one is inserted, you are asked to give a mark letter or number to be put with the symbol.



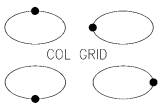
These three are for labeling a section, detail or elevation. When one is inserted, you are asked to give a mark letter or number a sheet number and a reference number to be put with the symbol.

These two blocks are to be used with the block below. Insert the circular block below with the triangular points first and then insert one of these flags, attaching it to the endpoint of one of the triangular points. If you do not want it touching the circular block you can move it after insertion. This will keep it lined up with the circular block.

This block is for cutting a section and giving the section number and sheet number. This block is a two part block. The outer circle is the first part. You may rotate it to any angle. The second part is the mark and sheet number. It will always be at a rotation angle of 0.

This is the north arrow. The arrow may be rotated to any angle but the "N" will always be at a rotation angle of 0.

This is a bubble for column line identification. You are prompted for the Grid mark. The mark will automatically be placed in the center of the bubble. Once inserted, the bubble may be rotated to the desired direction but the text will have a rotation of 0. Snap mode is automatically set to end snap for easy insertion.



This bubble is for when the column line identification has at least 4 characters. The dots in the icons represent the insertion points. Rotation is always 0. You are prompted for the column mark. The mark will automatically be placed in the center of the bubble and will have a rotation of 0. Snap mode is automatically set to end snap for easy insertion.

This is a bubble for column line identification where you would have a mark and a reference. The mark will be placed above the reference and will automatically be placed in the center of the bubble at a rotation of 0. Snap mode is automatically set to end snap for easy insertion.

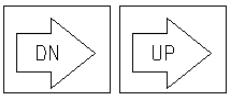
This is for drawing a single line end view of a "W" shape. The insertion point is the center of the web.

The AISC symbol is provided for your use for creating title blocks of customers that use the symbol. File: C:\autosd\sec\asd_aisc.dwg

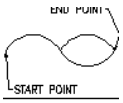
The AGC logo is provided for your use for creating title blocks of customers that use the symbol. File: C:\autosd\sec\agcoa.dwg

The AISC Certified Fabricator logo is provided for your use for creating title blocks of customers that use the symbol.
File: C:\autosd\sec\aisclogo-fab.dwg and C:\autosd\sec\aisclogo-fab.bmp

This is a typical revision symbol. You are asked for a number when it is inserted. You may enter a number or letter. The attribute name ASD_REV



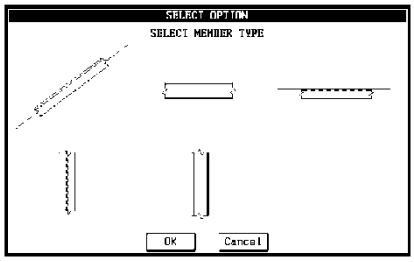
These are direction arrows. The arrow may be rotated to any angle but the “UP” and "DN" lettering will always be at a rotation angle of 0.



This is for drawing a "break” line in a round section. Pick the start and end points and the direction of the open end.

MEMBER

Pulldown: AutoSD > Erection Tools > Member
Toolbar: Erection Tools
Ribbon: AutoSD I > Erection Tools

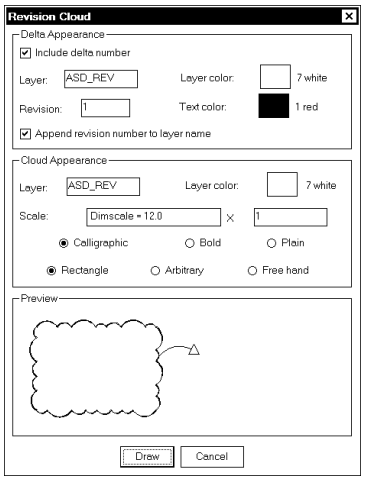


This program is for drawing a double line representation of a member on an existing single line. The program will break the line at the two points picked, erase that segment and draw the member picked from the icon menu. The scale is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command. The different member shapes are ANGLE, COLUMN FLANGE, COLUMN WEB, BEAM WEB and CHANNEL FLANGE.

REVISION CLOUD

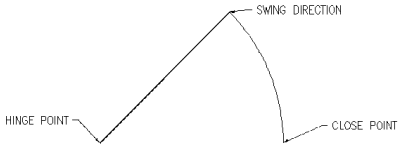
Pulldown: AutoSD > Draw > Rev. Cloud
Toolbar: Miscellaneous
Ribbon: AutoSD I > Miscellaneous

The revision cloud is for drawing a "cloud" around something. When drawing the cloud using the Arbitrary or Freehand option, move the cursor in a counter clockwise direction. In the Freehand option, the program will automatically close the cloud when the cursor is a short distance from the beginning. If the layer names are blank and "Append revision number to layer name" is unchecked, the cloud and delta will be drawn on the current layer. The delta will be the color of the current layer and not the color set in the configuration. Rectangular clouds will be drawn to the angle of the snap angle setting.



SAFETY GATE

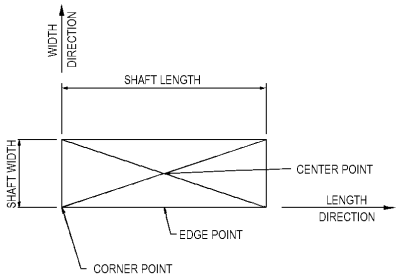
Pulldown: AutoSD > Erection Tools > Ladders/Gate
Ribbon: AutoSD I > Erection Tools



This program draws a safety gate using a polyline from three points picked and a given opening size. The scale is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.

SHAFT OPENING

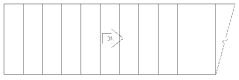
Pulldown: AutoSD > Erection Tools > Shaft Opening
Toolbar: Erection Tools
Ribbon: AutoSD I > Erection Tools



This program draws a shaft by supplying the shaft dimensions, start point and orientation. Shafts can be drawn from a corner, center or midpoint of an edge and at any angle. The scale is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.

STAIR PLAN

Pulldown: AutoSD > Stairs > Erection > Complete Plan
Pulldown: AutoSD > Stairs > Erection > Partial Plan
Ribbon: AutoSD II > Stairs

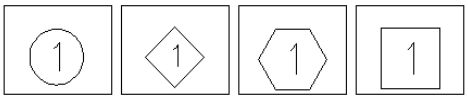


The partial plan is shown to the left with a "break" line at one end. The complete plan will look just like the partial except both ends will be square and will not have a "break" line at one end. The insertion point is one end of one stringer. The stair plan can be drawn in any direction and the scale is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.

`Location of end point of one stringer:` Pick first point.

`Stair direction:`	Pick second point or enter an angle.
`Side for other stringer:`	Pick a point.
`(FT.IISS) Width of stair:` **`3`**	
`(FT.IISS) Width of tread:` **`.1`**	
`Number of treads:` **`12`**	
`Direction of Stair (Down/Up):` **`D`**	
`Insertion point:`	Insertion point for direction indicator.
`Rotation: 1 found`	Rotate indicator.

TAGS



Toolbar: Miscellaneous
Ribbon: AutoSD I > Miscellaneous

This program is for inserting a circle, diamond, hexagon, square or triangle with an attribute in the center. These 4 symbols are tags and are provided for your use. They are similar to the revision symbol. The attribute name for each is listed below.

Symbol	Attribute Name	Symbol	Attribute Name
CIRCLE	ASD_CIR	DIAMOND	ASD_DMND
SQUARE	ASD_SQR	HEXAGON	ASD_HEX

TEXT DESCRIPTION

Pulldown: AutoSD > Erection Tools > Text Description
Ribbon: AutoSD I > Erection Tools

This program is for placing a text description parallel to a line on either side and centered at the point picked. Normally this would be the size of a beam, brace, column or some other member.

U-BLOCK SECTION VIEW

Pulldown: AutoSD > Erection Tools > U-Block Section View
Toolbar: Erection Tools
Ribbon: AutoSD I > Erection Tools

This program draws a cross section of 6", 8" and 12" block, (150mm, 200m and 310mm block). The blocks are drawn 7 5/8", (193.7mm), tall. The scale is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.

WOOD END VIEW

Pulldown: AutoSD > Erection Tools > Wood View
Toolbar: Erection Tools
Ribbon: AutoSD I > Erection Tools

This program draws the cross section of wood. Input the nominal size and the program will draw it to actual size. You enter the nominal width, nominal thickness, start point and orientation. The end view can be drawn from a corner, center or midpoint of an edge and at any angle. The scale is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.

Chapter 3 - Draw

@ LINE

Pulldown: AutoSD > Draw > @ Line
Ribbon: AutoSD I > Utilities

This command is similar to AutoCAD's line command. It operates in much the same way by starting with a point and asking for your next point. To enter the next point you may pick a point or enter the distance with or without an angle. If no angle is given, the angle will be from the previous point to the cursor. The distance is entered in the FT.IISS format and is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command. To draw a line 2'-3 1/4" at an angle of 90^0 from the last point it would be entered in the following way.

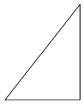
```
command: (FT.IISS) To point: @2.0304<90      enter
```

To draw a line using the cursor for direction:

```
command: (FT.IISS) To point: 2.0304          enter
```

2 PT VECTOR

Pulldown: AutoSD > Draw > 2 PT Vector
Ribbon: AutoSD I > Utilities



This program will draw the horizontal and vertical vectors of a sloping line. Pick the line and select side for the vectors. A horizontal and vertical line will be drawn from the end points of the selected line to form a triangle on the side of the sloping line that was selected for the vectors.

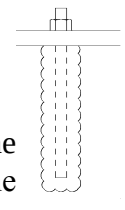
ADD STUDS

Pulldown: AutoSD > Add Studs
Toolbar: Detail
Ribbon: AutoSD II > Detail

This program will draw a side view or top view of any size threaded or headed stud along a selected line, with or without dimensions. The scale of the studs are affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.

ADHESIVE ANCHOR

Pulldown: AutoSD > Detail > Adhesive Anchor
Toolbar: Detail
Ribbon: AutoSD II > Detail

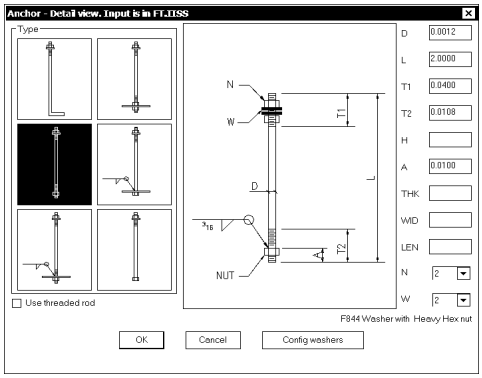


This program will draw a side view of any size bolt. The head will be solid lines; the shaft will be hidden lines with a "cloud" around the shaft representing the hole in the

concrete. The insertion point is the underside of the nut. The second point is the face of the concrete. The last point is the end of bolt inside the concrete. The first and last points determine the rotation. The scale is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.

ANCHOR BOLT

Pulldown: AutoSD > Detail > Anchor Bolt
Toolbar: Detail
Ribbon: AutoSD II > Detail



This program will detail six different types of anchor bolts. The insertion point is the top of the bolt. The description and type of washer used is set with the "Config washers" button. Select a washer and nut type for each anchor diameter. Washers can be hardened, non-hardened or plate. If a plate is used you also enter the thickness and width for a square plate. Threaded rod is an option for three of the types. The scale is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.

ARCHED TEXT

Pulldown: AutoSD > Arched Text
Ribbon: AutoSD I > Utilities

Arched text will produce text about a curve. Some examples are the AISC symbol and the slide in the ARCS program.

Command: Pick [AutoSD or AutoSD I > Utilities] > Arched text

```
Command: ATEXT
Pick radius point: 60,40
Pick middle point of text: 60,7
Text height: 1.5
Is base of text towards radius point <Y>:
Enter
Text: THIS IS ARCHED TEXT
Command:
```

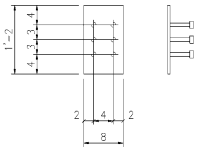
The text is written about an arch.

ASSEMBLY MARKS

Pulldown: AutoSD > Shop bill > Assembly Marks
Toolbar: Miscellaneous
Ribbon: AutoSD I > Shop bill

This is used to place assembly marks on your drawing. It works the same as the LEADER command except it keeps track of marks for plates, angles, bars, rods, channels, and miscellaneous marks. You can choose to have notes such as (NS), (FS), (BS), (CTR'D), R, L, and R/L automatically added to mark. You can increment the mark each time or repeat the same mark multiple times. The CNC option will allow you to make a DSTV file of a plain material piece.

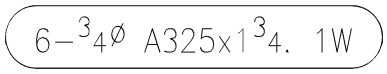
BEARING PLATE



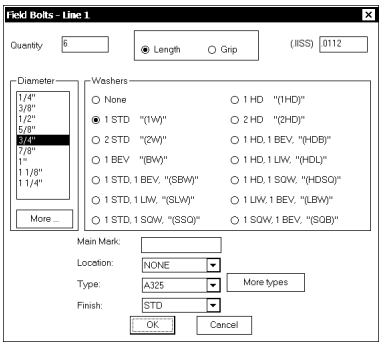
Pulldown: AutoSD > Detail > Bearing Plate
Toolbar: Detail
Ribbon: AutoSD II > Detail

This program will detail a plate with multiple rows and columns of holes or anchors on the far side and show the edge view of the plate. The scale is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command. Insertion is aided with the help of an outline box. Press the minus key on the numeric keypad while the outline box is displayed to reduce the length the picture will be drawn. The default size is to scale.

BOLT CIRCLE



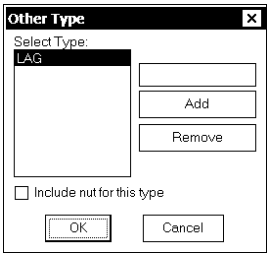
Pulldown: AutoSD > Draw > Bolt Circle
Toolbar: Miscellaneous
Ribbon: AutoSD I > Miscellaneous



This is a symbol that is used to locate the bolts for a field bolted connection. Select the proper options in the dialog box.

The choices for location are: NONE, LEFT, RIGHT, TOP, BOT. The choices for standard types are: A307, A325, A490, ADHESIVE, ANCHOR, EXP., LIB, SELF TAP, SLEEVE and WEDGE. ANCHOR, SELF TAP and SLEEVE are abbreviated ANCH, STAP and SLEV respectively.

You can add more diameters and types by picking the “More...” and “More types” buttons and entering the description in the edit box shown in the dialog box below. Pick ADD to add the description to the list on the left side which saves it. Put a check in the box below the list if a nut is included when listed in the bill of material, (for Types only). Pick REMOVE to delete a description from the list. Pick OK to add the list to the diameters or types in the main dialog box.



The choices for washers are:
(blank) for none
1W for 1 standard washer
1HD for 1 hardened washer

2W for 2 standard washers
2HD for 2 hardened washers
BW for beveled washer
SBW for 1 standard washer and 1 beveled washer
SLW for 1 standard washer and 1 load indicator washer
SSQ for 1 standard washer and 1 squirter washer
SBL for 1 standard washer, 1 beveled washer and 1 load indicator washer
HDB for 1 hardened washer and 1 beveled washer
HDL for 1 hardened washer and 1 load indicator washer
HDSQ for 1 hardened washer and 1 squirter washer
LBW for 1 load indicator and 1 beveled washer
SQB for 1 squirter washer and 1 beveled washer.
The description must be one of these choices for type for the extract program to recognize it.

The bolt circle uses attributes instead of text. The advantage of using attributes is the ability to extract the data to an ASCII file where it can be used by other programs. In order to extract information you must have a template file. The one shown below is for use with the Windows program EXTRACTW.EXE supplied with AutoSD Steel Detailing program and the bolt list program which is part of the CALCULATOR programs.

Attribute extract file \AUTOSD\BONUS\ASD_BLT.TXT
(The file FLDBOLT.TXT is used for drawings created with version 4.0 and earlier.
Do not modify these files. The information given below is for reference only.)

ASD_FIELDBOLT	C040000	(Description, 40 characters max.)
ASD_BLTMARK	C016000	(Main mark, 16 characters max.)
ASD_BLTLOCATE	C006000	(Location, which end, 6 characters max.)
ASD_BLTFINISH	C006000	(finish, i.e. black, galv, S.S. , 6 characters max.)

(The comments must not actually be part of the template file.) Each line has two fields. The first field is the tag name. The tag name is the name of one of the tags in the ASD_FLDBOLT block. The second field contains the format for the data for that tag. All format fields start with a "C" denoting it as a character field. The next three digits are the field width characters. The last three are number of decimal places. All character fields have "000" for the last three digits.

CAUTION: The first fields in the attribute extract template file MUST be spelled as shown above.

NOTE: Do not use "tab" characters when constructing the template file. If you want to align the columns for readability, do so with ordinary spaces.

BOLT EXPANSION

Pulldown: AutoSD > Detail > Bolt Expansion
Toolbar: Detail
Ribbon: AutoSD II > Detail

This program will draw a detailed side view of any size bolt. The head will be solid lines, the shaft will be hidden lines and tapered. The scale is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.

BOLLARD

Pulldown: AutoSD > Detail > Bollard
Toolbar: Detail
Ribbon: AutoSD II > Detail

This program will draw a pipe bollard, with or without anchors near the bottom. The scale is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.

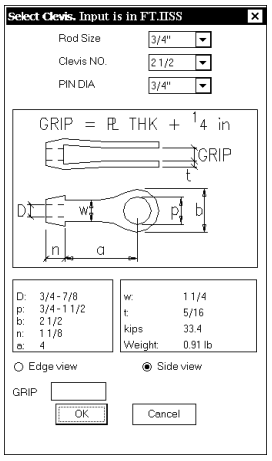
BOLT w/N&W

Pulldown: AutoSD > Detail > Bolt w/n&w
Ribbon: AutoSD II > Detail

This program will draw a detailed view of a variety of bolt types. You can draw a side view or end view of the head or nut, with or without washers. There is also a 3D option. The scale is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.

CLEVIS

Pulldown: AutoSD > Draw > Clevis
Toolbar: Miscellaneous
Ribbon: AutoSD I > Miscellaneous



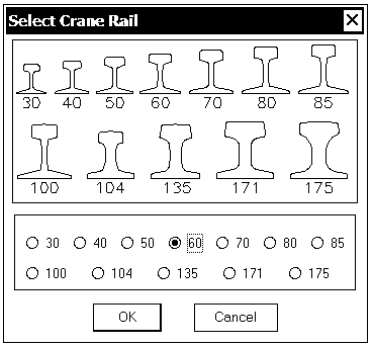
This will draw the side or edge view of a clevis. The insertion point is the center of the pin hole for the side view and the center of the pin midway of the grip for the edge view. The rotation is the angle from the insertion point to the center of the threaded end. The scale is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.

CLIP

Pulldown: AutoSD > Draw > Clip
Toolbar: Miscellaneous
Ribbon: AutoSD I > Miscellaneous

This is for drawing clip angles. The clips can have any number of rows, in either or both legs, be drawn at any angle, and with or without dimensions. They will be drawn to the scale set by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.

CRANE RAIL



Pulldown: AutoSD > Draw > Shapes > Next
Toolbar: Miscellaneous
Ribbon: AutoSD I > Miscellaneous

This is for drawing the end view of crane rails. The insertion point is the bottom center of the rail. These shapes will be drawn to the scale selected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.

DISTANCE

Pulldown: AutoSD > Tools > Distance
Ribbon: AutoSD I > Utilities

This is similar to AutoCAD's DISTANCE command. It returns the distance and angle between two points. The [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command will affect the lengths returned by this command.

DITTO

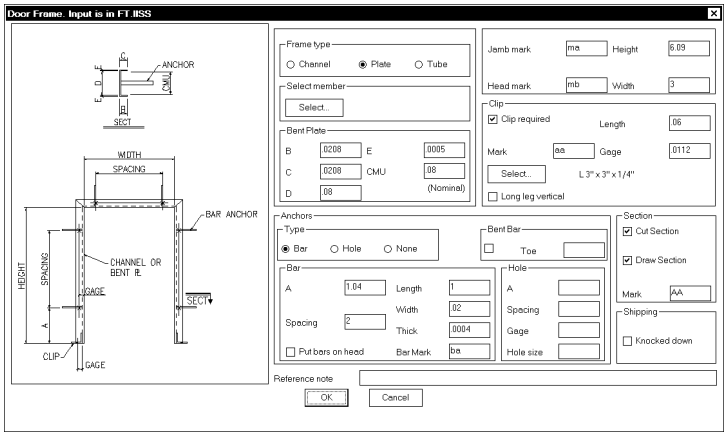
Pulldown: AutoSD > Draw > Ditto
Toolbar: Miscellaneous
Ribbon: AutoSD I > Miscellaneous

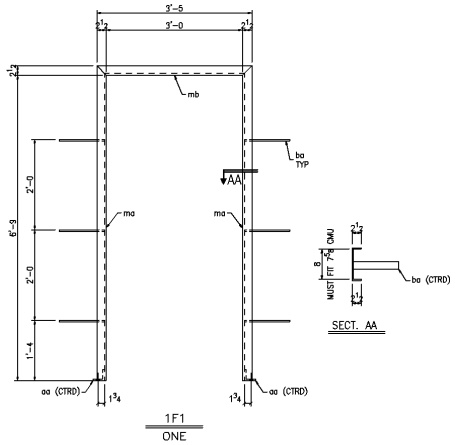
This draws a "do". It is used in the shop bill as a ditto symbol. The insertion point is the top of the "d". The second point is the bottom of the "d". It can be placed at any rotation.

DOOR FRAME

Pulldown: AutoSD > Detail > Door Frame
Toolbar: Detail
Ribbon: AutoSD II > Detail

This program will detail door frames made out of tubes, channels or bent plate. Anchor types are flat bar, holes in flanges (for channel or bent plate), or none. If you use a bar anchor it can be straight or bent. Section of jamb showing anchor can be cut and drawn or cut and not drawn. This is useful if you have other door frames on the same sheet with the same section detail. The clip at the bottom of the jamb is optional. Frames can be detailed as one piece or “knocked down” as two separate pieces. A reference not can be added that will be placed under the main mark.





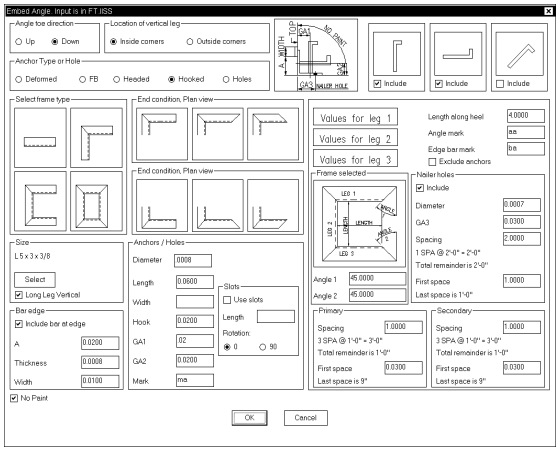
EL.

Pulldown: AutoSD > EL (vert)
Pulldown: AutoSD > EL (horiz)
Ribbon: AutoSD I > Miscellaneous

This command makes it easy to put an elevation dimension or text. After picking the insertion point for the arrow placement enter the elevation in the FT.IISS format or type "T" and press enter to enter a text string. One command is for placing it vertically; the other is for placing it horizontally.

EMBED FRAME

Pulldown: AutoSD > Detail > Embed Frame
Toolbar: Detail
Ribbon: AutoSD II > Detail



This program will detail a single member or a frame consisting of two, three or four members. The members may be angles or channels. The member lengths for a frame will be drawn to scale. Insertion is aided with the help of an outline box. If the selection is for a single member, press the minus key on the numeric keypad while the outline box is displayed to reduce the length the picture will be drawn. The default is to draw the length to scale and cannot be drawn longer.
Anchors can be a deformed rod, FB, headed stud, hooked rod or holes. Nailer holes can be included with anchors. You can also have an edge bar on top of the member if the member is an angle and is toed down or a channel. The program will automatically calculate the number of studs or holes that will fit a given length. The default for the first space is the same as the last space.

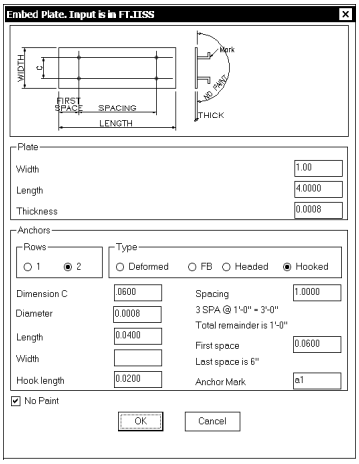
Example: Overall length = 20'-0, Spacing = 1'-6 returns:

```
13 SPA @ 1'-6 = 19'-6          Total remainder is 6"
First Space 0.03
Last space is 3"
```

If 3" is too small for the first space you can increase it and the number of spaces and the remainder will be re calculated. Embed angle can have one hole by making Hole spacing and First space the same dimension and at the center of the angle. The length must be drawn to scale for this option.

EMBED PLATE

Pulldown: AutoSD > Detail > Embed Plate
Toolbar: Detail
Ribbon: AutoSD II > Detail



This program will detail an embed plate. Insertion is aided with the help of an outline box. Press the minus key on the numeric keypad while the outline box is displayed to reduce the length the picture will be drawn. The default is to draw the length to scale and cannot be drawn longer.
Anchors can be deformed rod, FB headed stud, bent rod. The program will automatically calculate the number of studs that will fit a given length. The default for the first space is the same as the last space.

EXTRACT BOLTS (All Drawings)

Pulldown: AutoSD > Extract > Bolts All drawings
Ribbon: AutoSD I > Shop bill

Use this command to extract the field bolts from all drawings in a selected folder. You select one file in the folder in order to select the entire folder. It will create a file for each drawing with the same name as the drawing but with a ".blt" extension and puts the file in the same folder as the drawing. The file will contain a list bolts created with the Bolt Circle program.

EXTRACT BOLTS (Current Drawing)

Pulldown: AutoSD > Extract > Bolts Current drawing
Ribbon: AutoSD I > Shop bill

Use this command to extract the bolts from the current drawing. The default output file name for EXTRACT BOLTS is the drawing name with a TXT extension and is located in the same folder as the drawing. The program EXTRACTW.EXE is automatically run to convert the txt file to a blt file that can be used by the BOLT LIST program. You must purchase the CALCULATOR programs to get the BOLT LIST program. After the blt file is created the txt file is deleted.

Warning - AutoCAD 14 and BricsCAD only: If the file name is longer than eight characters, only the first six will be used to create the ".blt" file and the name will end with "~1". For example; the extracted file name 1998-402-09.txt will become 1998-4~1.blt. This could be a problem if the first six characters of two or more drawing names are the same because only one bolt list file will be created.

FIND TEXT STRING

Pulldown: AutoSD > Tools
Ribbon: AutoSD I > Utilities

This is used to find a given text string in the selected text. A "hand" pointer is placed on the text when a match is found. The pointer is on a layer called POINTER.

FLG HOLE

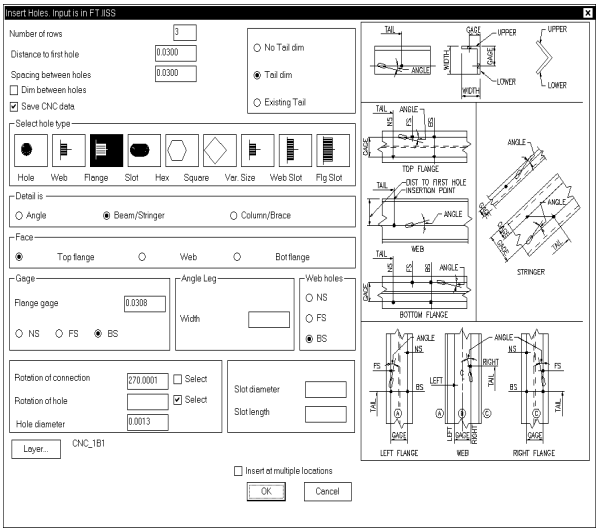
Pulldown: AutoSD > Draw > Flg Hole
Toolbar: Miscellaneous
Ribbon: AutoSD I > Miscellaneous

This is used for putting holes in the flange of a beam or other member where the thickness of material is drawn as 3/4" thick. It will put only one hole at a time. The scale is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.
If you have configured your customer to save CNC data you will be asked to select the tail dimension. If there is no tail dimension then select any line. The CNC editor will pop up for you to verify the gage, hole size, layer and tail dimension. Change it if needed, pick Apply and OK.

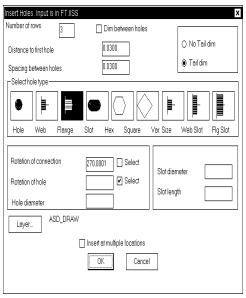
HOLES

Pulldown: AutoSD > Draw > Holes
Toolbar: Miscellaneous
Ribbon: AutoSD I > Miscellaneous

If you have configured your customer to save CNC data you will get the dialog box shown to the right. If not you will get the smaller one shown below right. More information is required for CNC data. If you have configured your customer to save CNC data but do not need it for this group of holes, uncheck the box "Save CNC data".



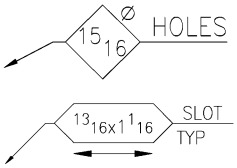
Edge view of web holes will not have CNC data attached to them.



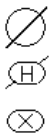
Solid filed holes with CNC data attached will be created from a single polyline. Holes without CNC data attached will be made using several lines close together so you can tell if you are saving CNC data with a hole just by looking at it.

Select whether the holes are for a beam or column. If the detail is a beam, brace, loose angle or anything other than a column then you would select Beam. It must be a column for you to select the Column option.

The HOLES command can draw a round hole, a 1/2" x 3/4" rectangular, 3/4" x 3/4" square, slots of the size you input and a hex or square head bolt. The 1/2" x 3/4" hole is used in edge view of clip angle legs or webs and the 3/4" x 3/4" hole is used in edge view of flanges. The “Var Size” hole will draw a rectangular hole of the size entered and the thickness is set by picking the start point and end point. The holes will be drawn to the scale set by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command. Select the options you want and the layer you want the objects drawn on and pick OK.



HOLE SIZE
SLOT SIZE



Pulldown: AutoSD > Draw > Hole Size
Pulldown: AutoSD > Draw > Slot Size w/Leader
Pulldown: AutoSD > Draw > Slot Size wo/Leader
Toolbar: Miscellaneous
Ribbon: AutoSD I > Miscellaneous

These are for labeling hole and slot sizes. There are two slot commands; one with a leader and one without. The insertion point for the one without the leader is the left end of the symbol. For the hole and slot command with leader you start with the location for the arrow. The "NEAR" snap is automatically activated for this selection. Pick the next point for the leader line and continue picking points. Press Enter after selecting the last point. If the first leader segment crosses any lines you will be asked if you want to select lines to omit arrowheads. If you answer No, arrows will be put at each intersection between the first leader segment and each line it crosses. If you want to select lines to omit arrowheads you should use "crossing" to select them in case there are two lines in the same location, (one on top of the other). The symbol is placed in the drawing at the end of the last leader segment. After entering the slot size you are prompted for a note. This note is placed under the word "slot" as shown in the example to the left. To leave the note blank press the space bar. After the symbol is completed you are prompted for <Rotation angle>/Reference: for the symbol. You may pick a point or enter an angle. If this is a round hole and you are saving CNC data you are prompted to "Select objects". You should select all the holes that this symbol references. The hole size will be associated with the holes so if you change the hole size the CNC data will be automatically updated.

LIB MNU

LIBRARY MENUS

Pulldown: AutoSD > Insert > Library Menus
Ribbon: AutoSD I > Utilities

Library menu is an icon menu that will display your blocks using icons allowing you to insert your block from the library by selecting the icon of the block. This gives you a visual reference to select from. The category SCALED will insert the block at the scale set by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command. If the block was saved in a drawing set up to a scale of 3/4" = 1'-0 and you are inserting it into a drawing set up to a scale of 1/8" = 1'-0 set the scale to 3/4" = 1'-0 to have it inserted the same. Select the icon picture and select OK. Enter “I” to insert or “D” to delete the block from the menu. The menu can contain an unlimited number of blocks. There is a "Next" button for advancing one menu at a time and a "Previous" button for backing up one menu at a time and a "Go To" button where you can go directly to the menu number you enter.

ADD TO LIBRARY

1	2	3
4	5	6
7	8	9

Pulldown: AutoSD > Insert > Add to Library
Ribbon: AutoSD I > Utilities

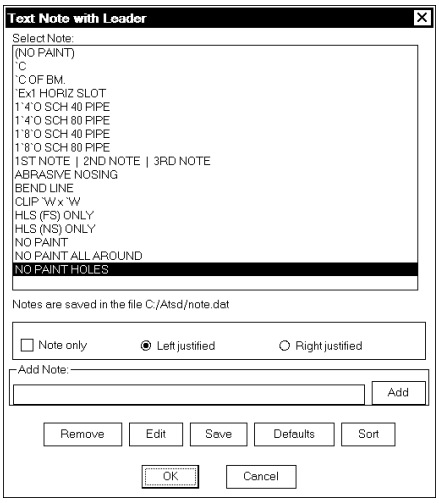
You can add new or existing blocks to the library menu by selecting "Add to library" from the AutoSD > Insert pulldown menu. You should be zoomed in on the objects that you want to add to the library. What shows on your display screen is what will be shown in the icon menu. The icons in each library are numbered starting with number 1 in the upper left corner and counting from left to right, top to bottom. There are 10 categories with an unlimited number of libraries in each category and 9 items in each library. The categories are: Beam, Brace, Column, Erection, Ladder, Miscellaneous, Rail, Scaled, Stair and Views.

CLEAN SLIDES

Pulldown: AutoSD > Insert > Clean Slides
Ribbon: AutoSD I > Utilities

This program will insert the blocks of the selected category one at a time, center the block, zoom to the extents of the block and make another slide. This will "clean" any garbage that may have been in the slide from surrounding lines when it was first made. This program will change only the slide picture that is displayed in the icon menu and will not affect the block itself. The program will erase all entities in the current drawing, therefore, you should start a new drawing to use this program and then quit the drawing when it is finished.

LEADER NOTES



Pulldown: AutoSD > Draw > Leader notes
Ribbon: AutoSD I > Dimension

This is for adding standard notes using a leader line. The notes available in the list box can be added to, removed, edited and reset back to the original defaults. Multi-line notes can be created by using the "|" character to separate the notes as shown in the last example in the list.

ADD NOTE Select the box under Add Note:. Enter the text for your note and pick Add.
REMOVE Select a note and pick Remove to delete the note.
EDIT Select a note and pick Edit. The note is now displayed in the edit box at the top. Make the changes in this box and pick Save to save it back to the list. Selecting Save at this time only saves the changes to the list. It does not save the list to the file. You will need to pick Save a second time to save to the file. If you decide not to save the edited changes back to the list just pick any note in the list to cancel the edit function and clear the edit box.

SAVE Saves the changes made to the list back to note.dat.
DEFAULT This will replace the list with the original list from note.def..
SORT This will put the list in alphabetical order.

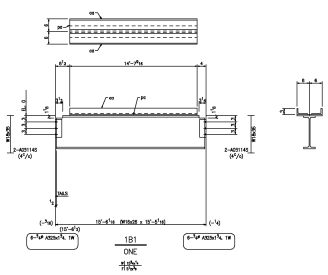
LINTEL (loose)

Pulldown: AutoSD > Detail > Lintel (loose)
Toolbar: Detail
Ribbon: AutoSD II > Detail

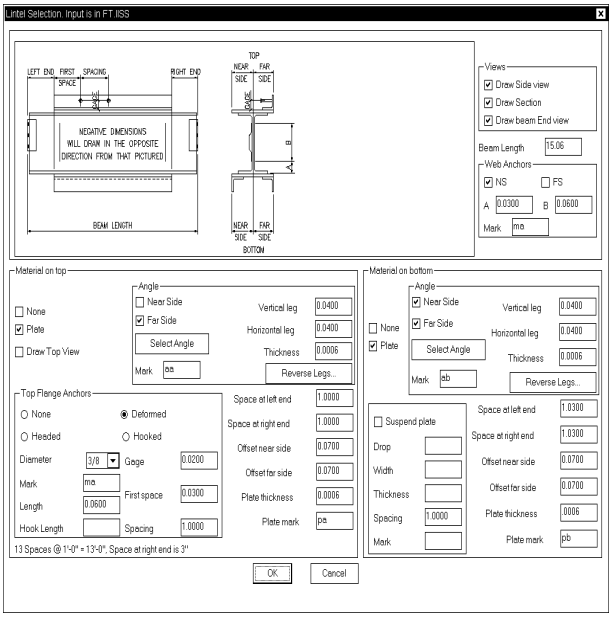
This program draws a lintel with a main shipping mark using 1 or 2 angles attached to a plate.

LINTEL (shop attached)

Pulldown: AutoSD > Detail > Lintel (shop attached)
Toolbar: Detail
Ribbon: AutoSD II > Detail



This program draws a lintel and/or masonry anchors on an existing beam detail. The lintel can be on the top flange, bottom flange or both. It can be a plate or angle or both. Angles can be near side, far side or both sides. You can draw any combination of side view, top view and end view. The end view can be on an existing end view of a beam or you can draw the end view of the beam too. The top view can only be drawn if a lintel is on the top flange. If there is a plate on the bottom flange it can be suspended below the beam or welded directly to the flange.



The beam length is required only if the side view is drawn. The lintel and anchor material description will be put in the shop bill only if the side view is drawn. You enter the beam length as the length shown on the detail from dimension line to dimension line, not the cut length or work length. The space at left and right end is entered as the distance from the dimension line on the beam to the end of the lintel. Enter a negative dimension if the lintel extends beyond the end of the beam.

To enter an angle size you can pick the "Select Angle" button or enter the leg size and thickness directly in the three boxes to the right of this button. If you need to swap the vertical and horizontal leg sizes just pick the "Reverse Legs" button.

Top flange anchors will be placed horizontally on one angle. If no angle is used, the anchors will be placed vertically on the top flange.

Enter values in all boxes that are not disabled by picking the box and typing the value needed using the FT.IISS format of input.

LIST FIELD BOLTS in SHOP BILL

Pulldown: AutoSD > List Field Bolts in Shop Bill
Ribbon: AutoSD I > Shop bill

This command will total, summarize and list the summary of field bolts in the shop bill giving a total of each different bolt, nut and washer listed in the current drawing.

MEASURE 2D LINE

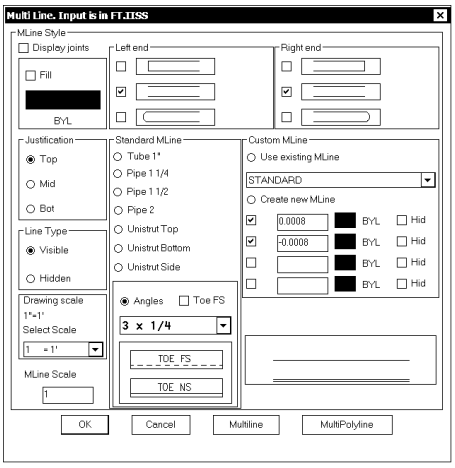
Pulldown: AutoSD > Tools > Measure 2D Line
Ribbon: AutoSD I > Utilities

This command works the same as AutoCAD's Measure command except that the spacing is controlled by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command. The command places point objects or blocks on a line at measured intervals using the FT.IISS format of input.

MULTILINE STYLE

Pulldown: AutoSD > Format > MultiLine Style
Toolbar: Miscellaneous
Ribbon: AutoSD I > Utilities

Create preset and custom MultiLines.
Display joints: Select this to draw a line at each joint.
Fill: Select this to fill the mline with a solid color.
Justification: Sets the location of the end points.
Line Type: The hidden option makes all lines hidden.
Select Scale: Sets the scale the mline will be drawn.
MLine Scale: Applies a scale factor to the mline.
Left end & Right end: Sets the closure style for the ends.
Standard MLine: Select to draw a preset mline.
Angles: Select an angle size for a preset mline.
Custom MLine: Use an existing mline by selecting it from the list or create a new mline. Selecting a mline from the list will populate all boxes so you can see the properties of the line. If you edit the boxes below "Create new MLine" you must select "Create new MLine" for those changes to be used.



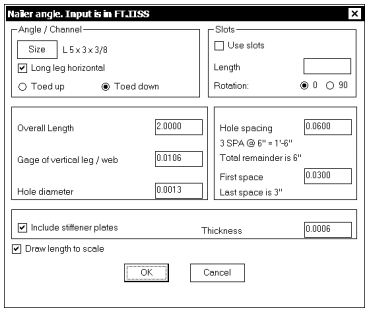
A sample mline using the options selected will be shown in the lower right corner of the Custom MLine section.

Buttons:
OK: will close the dialog box, create a new MLine and set the new MLine current. You can use the mline command or pick MultiLine from the AutoCAD menu to draw the line.
Cancel: will close the dialog box and void all changes.
Multiline: will close the dialog box, create a new MLine, set the new MLine current and start the mline command. After finishing you can press the enter key to open the dialog box.
MultiPolyline: same as the Multiline button but the lines will be drawn using polylines. (Does not work in AutoCAD 14.)

You can use the Tube 1" MLine and the MLine Scale to create any size pipe or tube mline. If you need a 1 ¾" tube, use a MLine Scale of 1.75 and the Tube 1" standard MLine.

NAILER ANGLE / Chanel

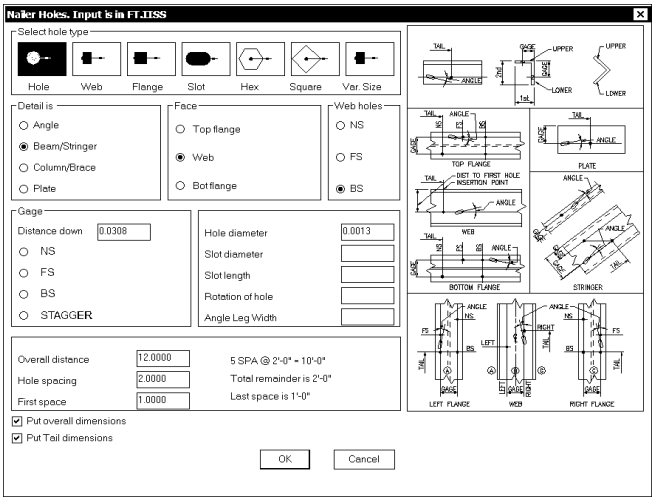
Pulldown: AutoSD > Detail > Nailer Angle
Toolbar: Detail
Ribbon: AutoSD II > Detail



This program will detail an angle that is to be shipped loose with holes or slots in one leg. The angle length may be drawn to scale. Insertion is aided with the help of an outline box. Press the minus key on the numeric keypad while the outline box is displayed to reduce the length the picture will be drawn. The program will automatically calculate the number of holes that will fit a given length. The default for the first space is the same as the last space.

NAILER HOLES

Pulldown: AutoSD > Nailer Holes
Toolbar: Detail
Ribbon: AutoSD II > Detail



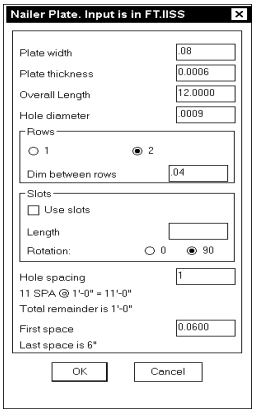
This command will automatically insert holes on a selected line in the web, flange or angle leg and optionally dimension the spacing and put tail dimensions to each hole.

The length of the first and last spaces will be drawn to scale. The distance between the first and last hole will be divided evenly.

Two different dialog boxes are used. One for CNC and another if you are not saving CNC data. The dialog box shown to the left is for CNC.

NAILER PLATE

Pulldown: AutoSD > Detail > Nailer Plate
Toolbar: Detail
Ribbon: AutoSD II > Detail



This program will detail a plate that is to be shipped loose with one line of holes or slots. The plate length may be drawn to scale. Insertion is aided with the help of an outline box. Press the minus key on the numeric keypad while the outline box is displayed to reduce the length the picture will be drawn. The program will automatically calculate the number of holes that will fit a given length. The default for the first space is the same as the last space.

OBJECT INFO

Pulldown: AutoSD > Tools > Object Info
Ribbon: AutoSD I > Utilities

This is similar to AutoCAD's LIST command. If you pick a line you will be given the length, bevel, angle of the bevel, horizontal vector, vertical vector and layer. The [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command will affect the lengths returned by this command.

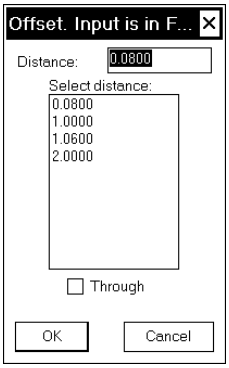
OFFSET

Pulldown: AutoSD > Modify > Offset
Toolbar: Modify
Ribbon: AutoSD I > Modify

This command works the same as AutoCAD's except that the distance is controlled by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command. By changing the scale of the drawing with the SCALE OF DRAWING command you can use OFFSET to draw to different scales. This offset command uses the FT.IISS format of input.

OFFSET w/Select (Icon has yellow background)

Pulldown: AutoSD > Modify > Offset w/select
Toolbar: Modify
Ribbon: AutoSD I > Modify



This command works the same as the offset command shown above except it uses a dialog box that allows you to select from the last 10 offset distances used or enter a new offset. The distance is controlled by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command. By changing the scale of the drawing with the SCALE OF DRAWING command you can use OFFSET to draw to different scales. This offset command uses the FT.IISS format of input.

OFFSET & CHANGE LINE TYPE

Pulldown: AutoSD > Modify > Offset & Change Linetype
Toolbar: Modify
Ribbon: AutoSD I > Modify

This command works the same as the offset command shown above except you are prompted on the first pick to select a linetype that the offset line will be changed to. This is useful in creating the hidden lines on pipe or tubes.

OSNAP

Menu: Ctrl > Right click or Shift > Right click
Toolbar: Object Snap

These are AutoCAD's commands for object snap and pick options. In order to draw a line from one line to another line you must use object snaps. If you don't, your lines will not be exactly where you want them to be. Some programs use object snaps and if you cancel the program in the middle of it you may find yourself locked into an object snap. If this happens pick NONE to get rid of it.

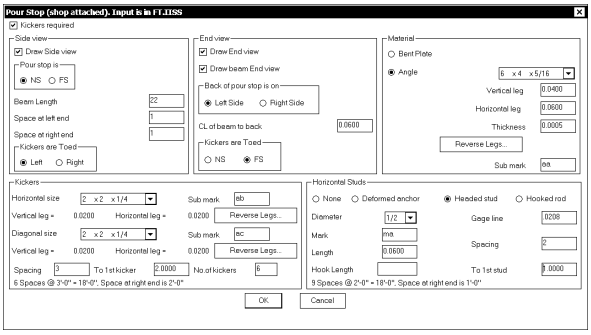
PIPE

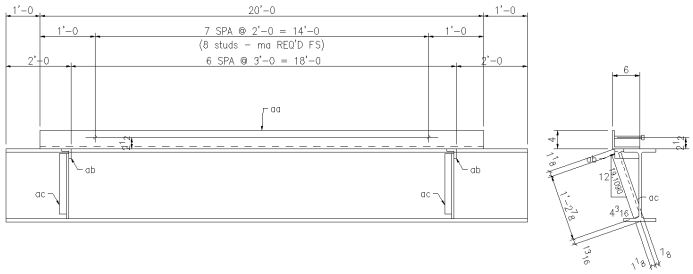
Pulldown: AutoSD > Draw > Pipe
Ribbon: AutoSD I > Miscellaneous

This program will convert a single polyline to a double line representation of a pipe with the option of drawing the inside wall. The scale is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.

POUR STOP (Shop Attached)

Pulldown: AutoSD > Detail > Pour Stop (shop attached)
Toolbar: Detail
Ribbon: AutoSD II > Detail



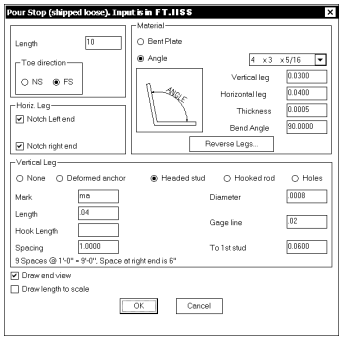


This draws a pour stop on top of a beam and/or a section view. The pour stop can be a bent plate or angle, toed near side or far side. You can draw just the side view, (on top of an existing beam), just the end view, (on top of an existing section or draw the section too), or draw both side view and end view. You can also have horizontal studs on the pour stop and kickers supporting the pour stop.

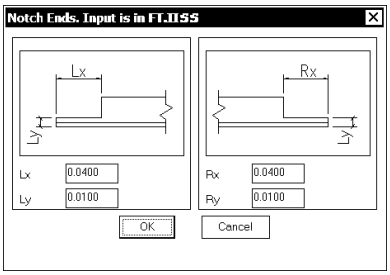
Under "Material" select either "Bent Plate" or "Angle". If you select Bent plate you will need to enter the size of the vertical and horizontal legs and thickness in the edit boxes under the drop down list for angles. If you select "Angle" these edit boxes will be filled in automatically from the angle size selected.
If you use the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command to set the scale to a larger scale the side view will be drawn to the scale of the drawing but the end view will be drawn to the larger scale.

POUR STOP (Loose)

Pulldown: AutoSD > Detail > Pour Stop (loose)
Toolbar: Detail
Ribbon: AutoSD II > Detail



This draws a pour stop as a main member. The pour stop can be a bent plate or angle, toed near side or far side. You can draw just the side view or draw both side view and end view.

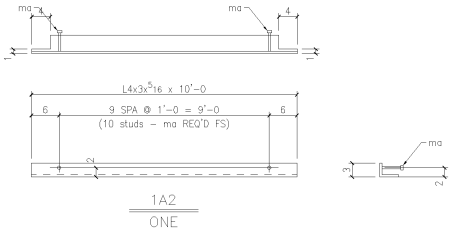


If a notch is required in either end in the horizontal leg, a top view will be drawn. You can also have horizontal studs on the pour stop.

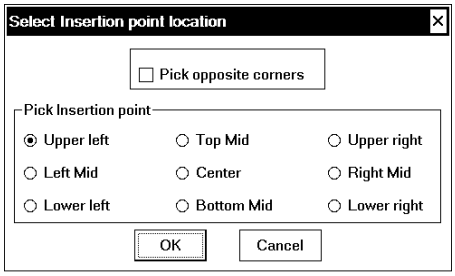
Under "Material" select either "Bent Plate" or "Angle". If you select Bent plate you will need to enter the size of the vertical and horizontal legs and thickness in the edit boxes under the drop down list for angles. If you select "Angle" these edit boxes will be filled in automatically from the angle size selected.

If you use the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command to set the scale to a larger scale the side view and end view will be drawn to the larger scale.

```
Main mark: 1A2
Are marks RIGHT and LEFT (Yes/No) <N>: Enter
            Quantity: 1
Shop bill information on mark 1A2
Place shop bill text on LINE <4>: Enter
```



RECTANGLE



Pulldown: AutoSD > Draw > Rectangle
Toolbar: Miscellaneous
Ribbon: AutoSD I > Miscellaneous

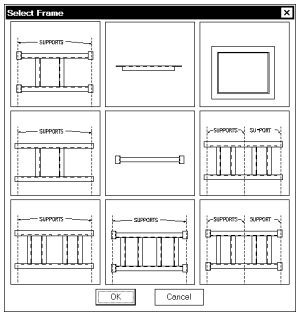
This draws a rectangle using a POLYLINE with a width of 0. The insertion point can be any corner, the midpoint of any edge, the center or two opposite corners.

ROTATE CROSSHAIRS

Pulldown: AutoSD > Tools > Rotate Cross hairs
Toolbar: Rotate Cross hairs
Ribbon: AutoSD I > Modify

```
Select option (0/45/Angle/Bevel/Control/Select) <S>:
```

This allows you to quickly set the angle of the cross hairs to match the angle of a line by picking the line if you choose the Select option. Other options are to input the Angle, Bevel, Controls, (horizontal & vertical vectors), 0 or 45 degrees.



ROOF FRAMES

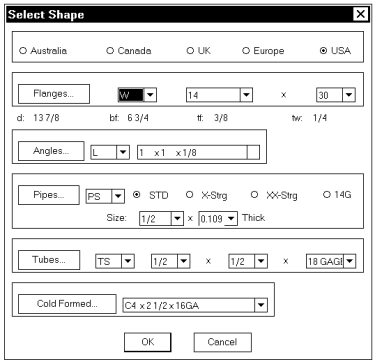
Pulldown: AutoSD > Detail > Roof frames
Toolbar: Detail

Ribbon: AutoSD II > Detail

There are nine different types of frames available as shown to the right. The top center and top right frames are drawn using angles only. The other frames can be drawn using angles or channels. The frames with vertical struts can be detailed with one or more vertical members as shown. Enter 0 for the location of each vertical member that is not required.

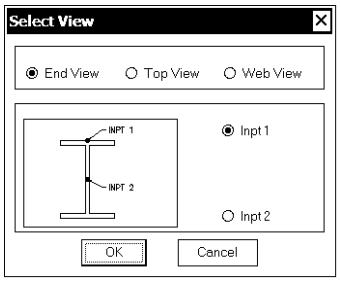
SCALE SHAPE

Pulldown: AutoSD > Draw > Shapes
Toolbar: Miscellaneous
Ribbon: AutoSD II > Miscellaneous l



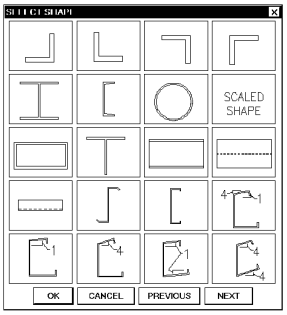
This is for drawing shapes to exact scale. The shapes available are "C", "HP", "M", "MC" "S", "W", “L”, “PS”, “TS” and “CF”. “CF” is for Cold Formed shapes made out of light gage metal. Flanges that slope will be drawn at 9o. These shapes will be drawn to the scale selected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command. A dialog box is used to select the shape. Just pick the type of member and size and then pick the OK button.

The three views that can be drawn are TOP VIEW, ENDVIEW and WEBVIEW.
The insertion point for each view is as follows:
END VIEW: At the top on the center line or at the middle of the web on the center line.
TOP VIEW: Each end on the center line.
WEB VIEW: Each end at the top



SHAPES

Pulldown: AutoSD > Draw > Shapes
Toolbar: Miscellaneous
Ribbon: AutoSD I > Miscellaneous



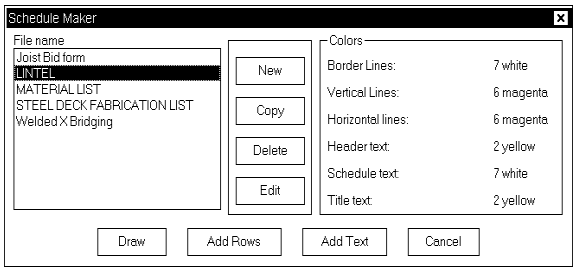
This menu is for drawing the end view, side view and top view of various shapes. This will draw the shapes with a flange and web thickness that will look attractive when plotted. The flange will be 3/4" thick and the web and angles will be 1/2" thick at a scale of 1"=1'. ZEE and CEE shapes are drawn 1/4” thick. *Scaled Shapes* will draw various shapes to exact scale and thickness.
ZEE and CEE sizes are listed by depth x width x thickness.

Wshape.dwg:

Drawing “Wshape.dwg”, located in your C:\autosd\sec folder, is a special dynamic block. Once inserted it may be changed to represent any size “W” shape. You must be using AutoCAD 2010 or later to use this block. The insertion point is the mid-point of the top flange. To change the size select the block and click on the triangle at the insertion point and select the depth and weight.

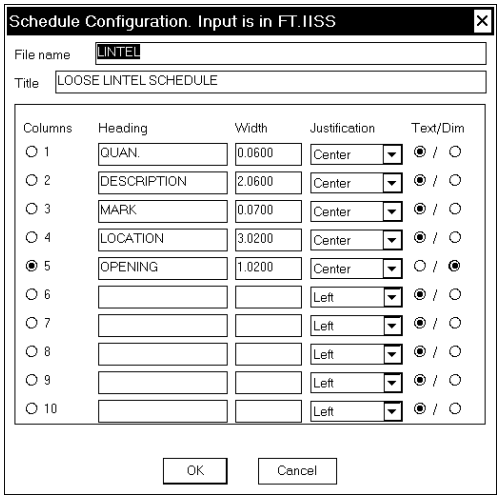
SCHEDULE MAKER

Pulldown: AutoSD > Draw > Schedule Maker
Ribbon: AutoSD I > Utilities

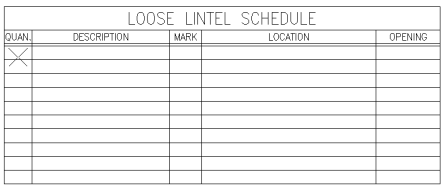


This is for creating, drawing and adding text to a schedule. Select the File name from the list and then pick the operation.

New: This will add a new schedule file to the list and open it for editing.
Copy: This will copy an existing schedule file and open it for editing.
Delete: This will delete the selected schedule file.
Edit: This will open the selected schedule file for editing.
Colors: Pick the color swath to change the color.
Draw: This will draw the selected schedule and prompt for text to be placed on the first line.
Add Rows: This is for adding rows to an existing schedule.
Add Text: This is for adding text to an existing schedule.



The dialog box shown to the left is for editing a schedule file. Enter a file name and the title that will be placed at the top of the schedule. You can have up to 10 columns with column 1 on the left side. Enter the text you want for the heading of each column. Multi-line headings can be created by using the "|" character to separate the headings. Enter the width of the column, select the justification for the text and pick if the input is for text or a dimension.



When selecting the insertion point for the schedule, an image of the outline of the schedule will be drawn to allow for easy placement. Press the “+” key to make the schedule taller. Press the “-“ key to reduce the height of the schedule. After the schedule is drawn you are prompted for the text for the first column. An “X” is drawn at the insertion point of the text. After the text is entered it is placed in the column and the “X” moves to the insertion point of the text for the next column and you are prompted for the text. After the last column is entered the “X” moves to the intersection of the next line down and the left border

of the schedule. You are given the options to Continue, Skip or eXit. The default is to continue. Pressing Enter moves the "X" to the insertion point of the text in the first column and you are prompted for the text. If you choose to Skip, the "X" will move down one line and the options to Continue, Skip or eXit will be repeated. Entering X at this prompt will exit the program.

STANDARD MTEXT

Pulldown: AutoSD > Draw > Standard Mtext
Toolbars: Miscellaneous
Ribbon: AutoSD I > Miscellaneous

This program will open a dialog box and give you three text sizes to choose from, 1/8", 3/16" and ¼". Pick the button for the size you want and pick OK. Select two corners and a dialog box for the mtext opens. You enter your text here as well as set the color.

TAIL

Pulldown: AutoSD > Draw > Tail to Here
Ribbon: AutoSD I > Miscellaneous

This places a block to indicate where dimensions are referenced from. You set which block you want to use in the customer configuration and the text that you want to use if you select the block with text. The block can be a target that looks like the picture to the left or text on an arrow.

TAIL or TAIL DIM

Pulldown: AutoSD > Dimension > Tail Dim
Toolbars: Dimension I
Ribbon: AutoSD I > Dimension

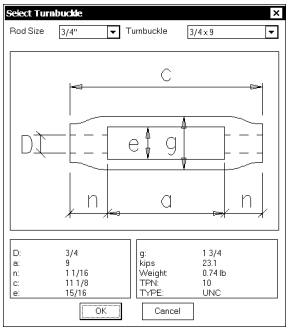
This Places a tail, or stub dimension with a line under it. If you use "Select Dim to Add" under AutoSD > Calculator, the default dimension will be the result. Input is in the FT.IISS format.

TEXT STRINGS

Pulldown: AutoSD > Draw > Text Strings
Ribbon: Not included

These are for placing common notes in response to a text prompt. Most of these are for the description of material for the shop bill such as pipe and grating. They will not work when using the text command.

TURNBUCKLE



Pulldown: AutoSD > Draw > Turnbuckle
Toolbar: Miscellaneous
Ribbon: AutoSD I > Miscellaneous

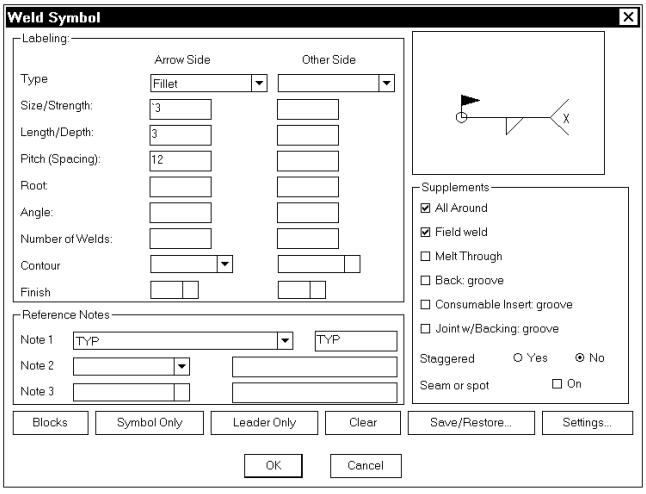
This will draw the side view of a turnbuckle. The insertion point is the center of the turnbuckle. The rotation is the angle from the insertion point to the center of the threaded end. The scale is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.

WEB HOLE

Pulldown: AutoSD > Draw > Web Hole
Toolbar: Miscellaneous
Ribbon: AutoSD I > Miscellaneous

This is used for putting holes in the web of a beam or other member where the thickness of material is drawn as 1/2" thick. It will put only one hole at a time. The scale is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.

WELD SYMBOLS



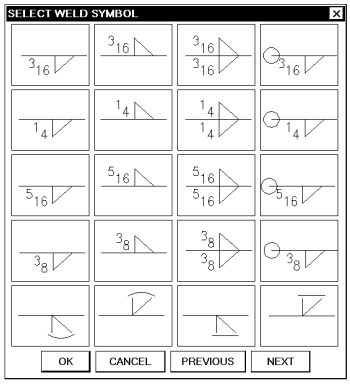
Pulldown: AutoSD > Draw > Weld Menu
Toolbar: Miscellaneous
Ribbon: AutoSD I > Weld

This program is for creating and placing weld symbols. This command uses a dialog box to select the weld symbols. The weld types include Fillet, Bevel, Bevel + Fillet, Bevel Flare, J, J + Fillet, Square, Square + Fillet, U, V, V-Butt w/Insert, V-Flare, V-T w/Insert, Plug or Slot, Seam, Spot or Projection, Stud and Surfacing. Select a weld type by clicking on the down arrow next to Type, then click on the weld type desired. The weld symbol displayed in the window will be updated automatically.

The properties available for each weld type vary. For example, Back, Consumable Insert and Joint w/Backing are only available when one of the grove welds is selected. Staggered is only available when Fillet is selected for arrow side and other side. You can place a complete weld symbol or add a leader, all around symbol, field weld symbol, or tail note to an existing weld symbol.

BLOCKS

Pulldown: AutoSD > Draw > Weld Blocks
Toolbar: Miscellaneous
Ribbon: AutoSD I > Weld



This allows you to place a simple weld block quickly or add an all-around, field weld or tail note to an existing weld symbol. You can add one of these blocks to an existing symbol by picking the "SYMBOL ONLY" icon from the second page of blocks and then select the block. This prevents the leader from being drawn again and will insert only the block. The insertion point is the left end of the block. "ENDPOINT" snap is automatically selected.

SYMBOL ONLY

This draws the symbol shown in the dialog box without adding a leader.

LEADER ONLY

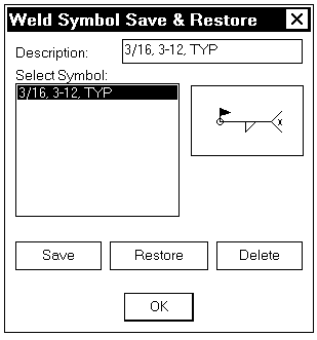
Pulldown: AutoSD > Draw > Weld Leader
Toolbar: Miscellaneous
Ribbon: AutoSD I > Weld

This draws only the leader. It is useful for adding more leaders to an existing weld symbol. Pick the location for the arrow. The "NEAR" snap is automatically activated for this selection. Pick the next point for the leader line and continue picking points as needed. The last point should be one end of the weld symbol. Press enter and you are then prompted for additional leader lines. Press enter for none. If you select to put additional leaders the near snap is activated and you pick where the leader arrow starts. Press enter and a line is drawn from the last pick point to the weld symbol. Press enter again to exit drawing leader lines.

CLEAR

Selecting the *Clear* button will clear all selections and labels.

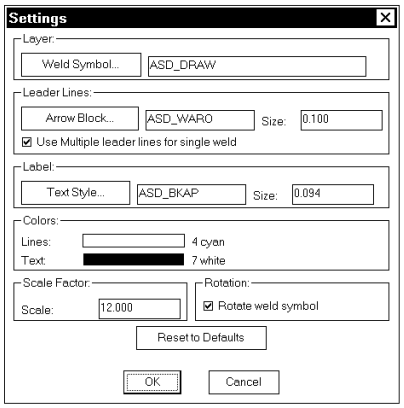
SAVE/RESTORE



This button opens another dialog box that allows you to save, delete or restore a weld symbol. The symbol in the main dialog window is shown in a window in this box. Click on the box to the right of *Description* to save this symbol. Type in a name for the symbol, press enter and click the *Save* button. The name will be displayed in the *Select Symbol* list box and the configuration will be saved in the file *weldsym.dat*. You can use any characters in the name and the name can be any length. If you have some names in the list box you can restore one of these by selecting it from the list box. The window is updated to display the selected symbol. Pick the *Restore* button and you are

returned to the main dialog box and the weld symbol settings will be set to those defined by the configuration you restored.

SETTINGS

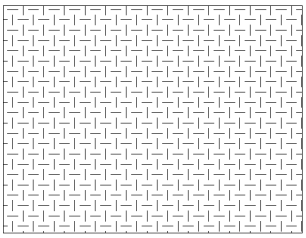


This button opens another dialog box that allows you to change the layer the symbol is drawn on, the block used for the arrow head, the style used for the text, the colors used for the lines and text, the size of the text, the scale factor for the symbol, whether to use multiple leaders or not and whether to rotate the symbol or not. The settings are saved to the file weld.cfg located in your autosd folder.

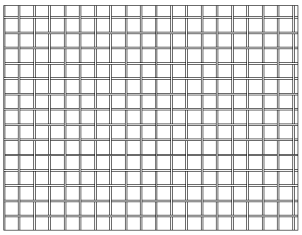
HATCH

AutoCAD command. Pulldown: Draw > Hatch

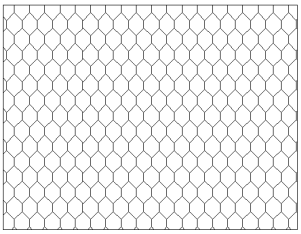
There are three custom hatch patterns installed with AutoSD. They are located in your autosd\Support folder. The names of the pattern files are checkpl.pat, mesh.pat and wire.pat. A sample of each is shown below.



Checkpl.pat



Mesh.pat



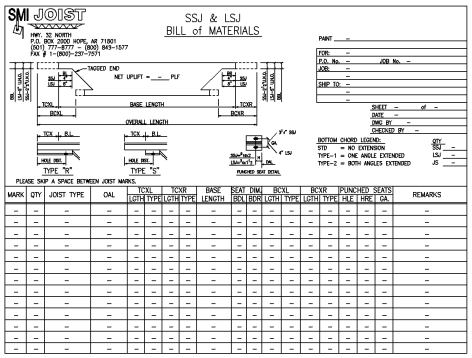
Wire.pat

You can use these with the bhatch command. Type Bhatch. Change the *Type* to *Custom* and click on the *Swatch* box select the pattern from the *Custom pattern* box.

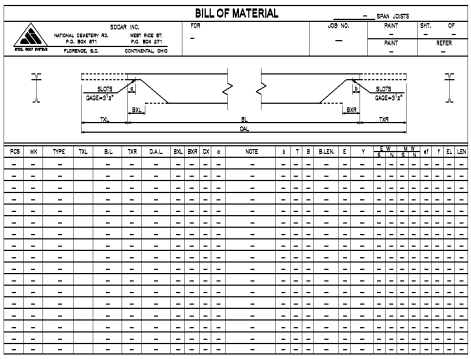
JOIST FORMS

Forms for Valley Joist can be downloaded from this web site.
http://www.valleyjoist.com/downloads/valleyjoisteast

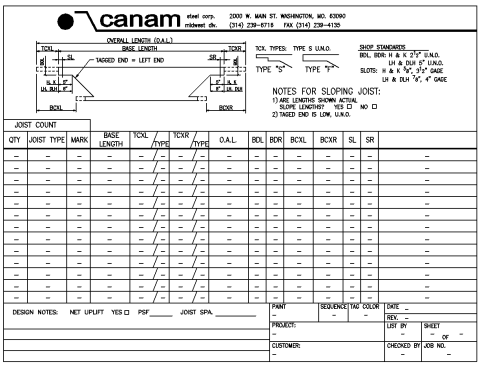
The following form drawings are located in your C:\autosd\bonus folder. The drawings are 8 1/2" x 11". The dash marks in each field can be changed using the Replace Text command to automatically position the text when filling out the form.



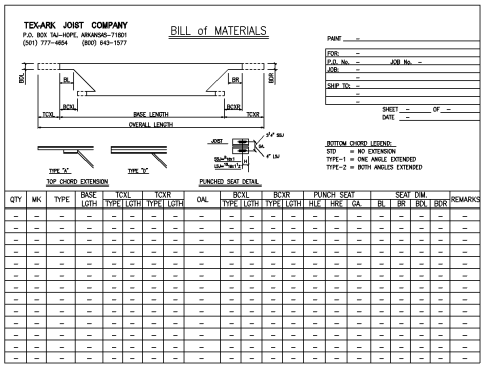
SMI JOIST.DWG



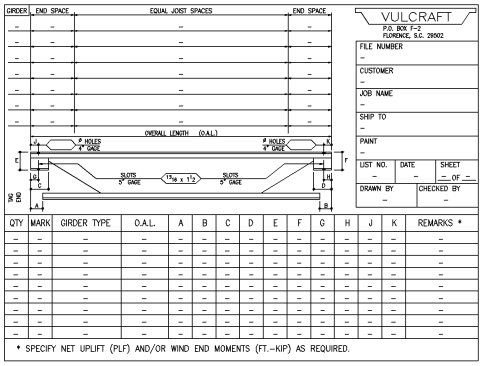
SOCAR JOIST.DWG



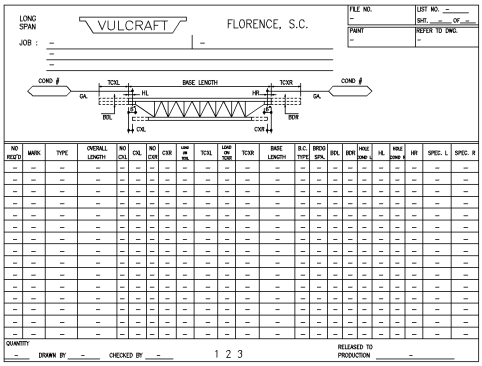
CANAM JOIST.DWG



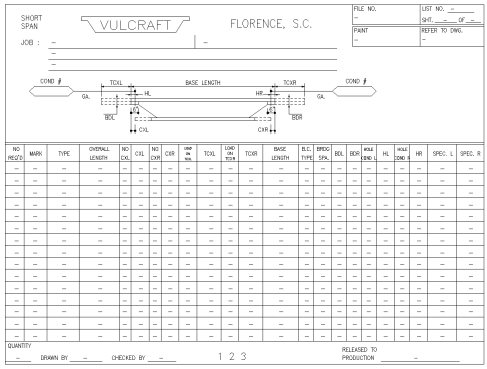
TEX-ARK JOIST.DWG



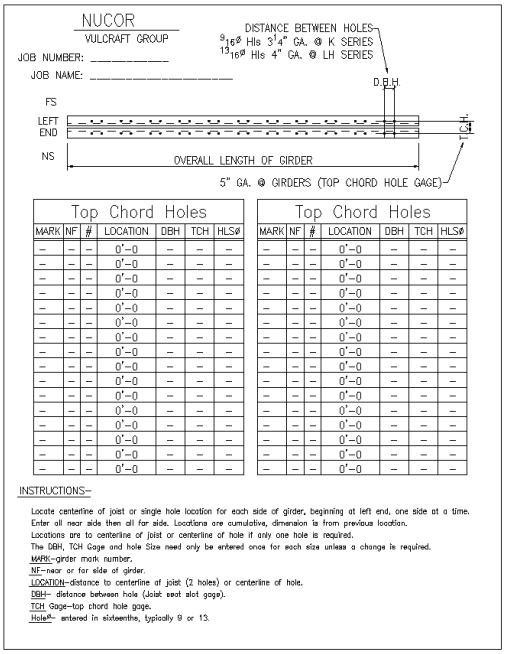
VULCRAFT GIRDER.DWG



VULCRAFT LONG SPAN.DWG



VULCRAFT SHORT SPAN.DWG



NUCHOR TOP CHORD HOLES.dwg

Chapter 4 - Edit

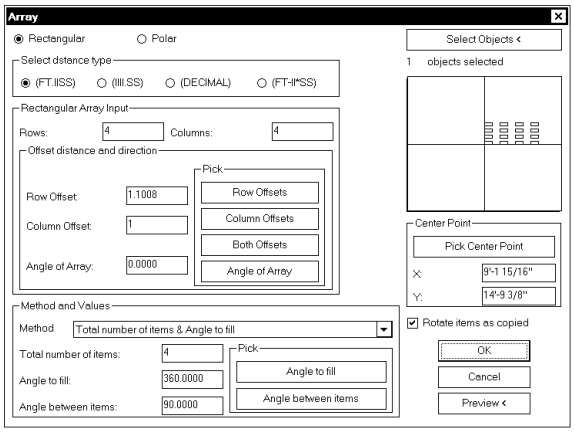
ADJUST DIMENSION

Pulldown: AutoSD > Modify > Adjust Dimension
Toolbar: Change
Ribbon: AutoSD I > Change

This command will change a dimension by the amount entered. All dimensions selected will be changed, either increased or decreased. It takes a number for input in the FT.IISS format.

ARRAY

Pulldown: AutoSD > Modify > Array
Toolbar: Modify
Ribbon: AutoSD I > Modify



The array command is similar to the standard AutoCAD array command. The exceptions are you can enter the row and column offsets in FT.IISS, IIII.SS, DECIMAL OR FT-II*SS format and these offsets are affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.
The PREVIEW button allows you to see the result before you make it final.

ARROWHEADS TO TICKS

Pulldown: AutoSD > Modify
Ribbon: AutoSD I > Change

This command will change all selected arrowheads to ticks.

BREAK LINE TYPE

Pulldown: AutoSD > Modify > Break > Break Line Type
Toolbar: Modify
Ribbon: AutoSD I > Modify

This will break a line at two selected points and then change the segment that was picked first to the line type selected from a dialog box. It is normally used to break a continuous line to make part of it hidden. Select the line where the hidden segment will be, then picked a point on each side of the selection point. Select HID or HIDDEN from the dialog box.

CHANGE TEXT

Pulldown: AutoSD > Modify > Change Text
Toolbar: Change
Ribbon: AutoSD I > Change

Change text is used to change part of a word or number by typing in the old string, the part you want to change, and then typing in the new string, what you want to change it to.
For example: To change "THIS BROWN COW" to "THIS BLACK COW" pick change text from tablet.

```
Command:
Select objects: 1 selected, 1 found.
Select objects: Enter
Old string: ROWN
New string: LACK
Changed 1 text lines.
Command:
```

Change Text command will change more than one string of text at a time. For example, if you are using the sheet number as part of the piece marks you can change as many as you can pick by typing in the old number and then the new number. Change Text also works on attributes like the main piece mark and bolt description but must be used with caution. It will change ALL attributes in a block where a match is found.

CHANGE TO PLINE

Pulldown: AutoSD > Modify > Change To Pline
Toolbar: Change
Ribbon: AutoSD I > Change

This command will change lines to polylines. It will also join lines and polylines together to form one continuous polyline. Polylines are used in hatching and for calculating area.

COLOR

Pulldown: AutoSD > Modify > Color
Toolbar: Change
Ribbon: AutoSD I > Change

This will change the color of the objects selected to the color selected and change the line weight to that of the new color if Line weights are enabled for the customer colors.

COLOR w/filter

Pulldown: AutoSD > Modify > Color > Color w/filter
Toolbar: Change
Ribbon: AutoSD I > Change

This will change the color of lines or text of one color to the color selected.

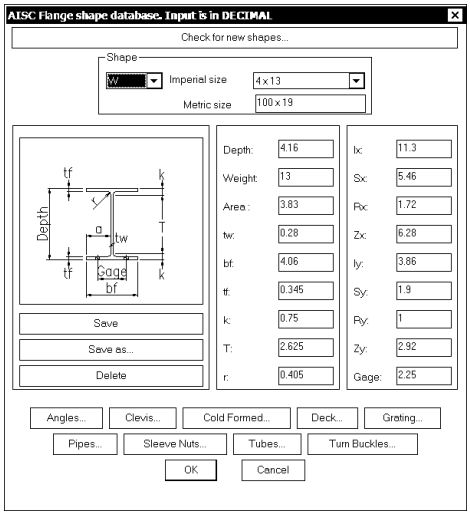
DDCOPY

Pulldown: AutoSD > Modify > DDcopy
Ribbon: AutoSD I > Modify

Copies objects from other layers to the current layer. It will not copy objects on the current layer.

DATABASE EDITOR

Pulldown: AutoSD > Modify > AISC database
Ribbon: AutoSD I > Modify



This program will allow you to edit all information in the AutoSD AISC database of US shapes. The example to the left shows the dialog box for editing "W" shapes but all dialog boxes work the same. First, select the type of shape then the size that you want to edit. This is selected from a drop down list. The metric equivalent is given in an edit box below the drop down list. Click on any edit box and make your change. All dimensional input is in decimal inches. After all editing is finished pick OK to save the changes back to the file. Picking Cancel will cause you to lose all changes for that shape type.

Save: This will save changes to the current size. Selecting a different size will also save any changes to the current size.

Save as: This will save the current size to a new size. All changes made before picking "Save as" will be saved to the new size and not the current size.

Delete: This will delete the current size from the database. If it exists in the original data base it can be retrieved otherwise it cannot.

If you are upgrading from an older version, your changes to the database will not be overwritten when you install the new version of AutoSD. The New version will install the database files in your C:\autosd\data\backup folder. The database files that you edit are saved in your C:\autosd\data folder. If any new shapes are added to the database on the installation CD, they will be listed in a dialog box that will allow you to add them to your database files when you use the AISC database editor for the first time. The editor only works on US shapes. It will not edit the data base files for shapes from other countries.

EDIT ATTRIBUTE

Pulldown: AutoSD > Modify > Attribute
Ribbon: AutoSD I > Modify

These commands are for editing attributes. Some places you will find attributes are in the title block and in section symbols.

CO LINE TYPE

Pulldown: AutoSD > Modify > Line Types
Toolbar: Change
Ribbon: AutoSD I > Change

These 8 commands will change the line type of the lines picked to the type selected. The line types are continuous, dashed, hidden, center, phantom, dot, dashdot and border.

LINE WEIGHT

Pulldown: AutoSD > Modify > Line weights
Toolbar: Change
Ribbon: AutoSD I > Change

This command will change the line weight of the line or text or both. The selection can be filtered by color or all selected can be changed. The line weights will be changed to the weights stored in your customer configuration for colors.

RESET DWG SCALE

Pulldown: AutoSD > Modify > Reset DWG Scale
Toolbar: Change
Ribbon: AutoSD I > Change

This command will reset the scale of the drawing back to the original scale and will place the scale in the left end of the status bar.

REP REPLACE DIMENSION

Pulldown: AutoSD > Modify > Replace Dimension
Toolbar: Change
Ribbon: AutoSD I > Change

This command will replace a dimension or a string of text with a dimension. It takes a number for input in the FT.IISS format or two points.

`Command:`	Pick Replace dimension
`Select objects: 1 selected, 1 found`	Pick text
`Select objects:` **`Enter`**	
`(FT.IISS) Spacing/Pick/Dimension:` **`10.0312`**	10'-3 3/4
`Command:`	Dimension is changed to 10'-3 3/4.

Enter “S” to enter number of spaces and length of space.

```
(FT.IISS) Spacing/Pick/Dimension: S
```

```
___SPA @____ =
Enter number of spaces: 2
(FT.IISS) Length of each space: .03          Result is "2 SPA @ 3 = 6"
```

Enter "P" to enter distance by picking two points.

```
(FT.IISS) Spacing/Pick/Dimension: P
Pick first point
Pick second point          Result is the scaled distance between the two points.
```

Using the "P" option the distance picked is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.

Dimensions can be put on the drawing more easily with AutoSD > Dimension > DIM. TEXT than with the TEXT or DTEXT command. It works like the text commands except the input is FT.IISS only.

REPLACE TEXT

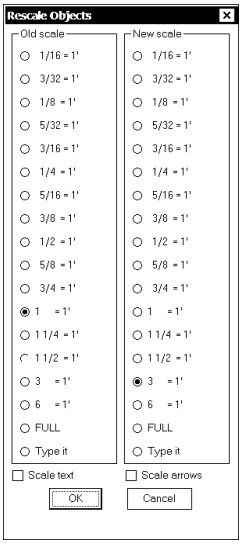
Pulldown: AutoSD > Modify > Replace Text
Toolbar: Change
Ribbon: AutoSD I > Change

Replace text is similar to change text except that you are asked only for the new text and all of the old text that was picked is replaced with the new text.

RELOCATE TEXT

Pulldown: AutoSD > Modify > Relocate Text
Toolbar: Change
Ribbon: AutoSD I > Change

Relocate text is for moving dimensions that have been mirrored and need to be moved back to the other side of the dimension line. Multiple selections are possible using window or crossing or select one at a time.



Scale of Drawing

Pulldown: AutoSD > Modify > Scale of Drawing
Toolbar: Change
Ribbon: AutoSD I > Change

The Scale of Drawing command will enable you to draw to a different scale in the same drawing. After picking Scale of Drawing you are asked to "Select the scale from the dialog box: ". Selecting a scale will change the drawing to the scale selected. The scale that you select will appear in the left end of the status bar.
Commands affected by the Scale of Drawing command are "Distance", "Object info", dimension commands and "Offset". This will enable you to

draw to different scales in the same drawing when using the OFFSET command.

SCALE OF OBJECTS

Pulldown: AutoSD > Modify > Scale Of Objects
Toolbar: Change
Ribbon: AutoSD I > Change

This is for changing objects from one scale to another. This command can be real useful if you draw or insert a section to one scale and decide that it needs to be bigger or smaller. If the check boxes for text and arrows are left unchecked, the size of text and arrows will remain unchanged after scaling. This function also works in Paperspace by changing the zoom scale of the selected viewport.

SCALE OF BLOCKS/SOLIDS/TEXT

Pulldown: AutoSD > Modify > Scale Of Blocks/Solids/Text
Toolbar: Change
Ribbon: AutoSD I > Change

This command is similar to the Scale of Drawing command except the base point is the insertion point of the solid, block or text. When you pick this command you are prompted for "Which items to scale (All/BLocks/Solids/Text)". Type the letter of the choice shown by the capitol letter and then select the objects. Arrow heads and ticks are solids. You are then asked to input a scale factor. If you enter 2, the items picked will be made twice as big and will be scaled out from their individual insertion points.

LAYERS

CHANGE OBJECT'S LAYER

Pulldown: AutoSD > Format > Change Object’s Layer
Ribbon: AutoSD I > Utilities

This will change all objects to the current layer.

FREEZE LAYER

Pulldown: AutoSD > Format > Freeze Layer
Ribbon: AutoSD I > Utilities

This will freeze all layers that belong to the objects you pick.

ISOLATE LAYER

Pulldown: AutoSD > Format > Isolate Layer
Ribbon: AutoSD I > Utilities

This will freeze all layers except the ones that belong to the objects you pick.

SET LAYER

Pulldown: AutoSD > Format > Set Layer
Ribbon: AutoSD I > Utilities

This will set the current layer to that of the object you pick.

THAW ALL LAYERS

Pulldown: AutoSD > Format > Thaw All Layers
Ribbon: AutoSD I > Utilities

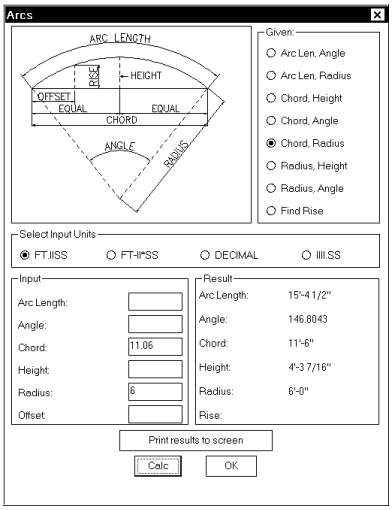
This will thaw all layers
.

Chapter 5 - Decimal & FT.IISS Calculators

There are several different programs of each type of calculator. The only difference between the two types is the type of input. The decimal calculator requires decimal numbers for input and the output will be in decimals. The FT.IISS calculator requires that numbers be input in the FT.IISS format.

ARCS

Pulldown: AutoSD > Calculator > Arcs
Toolbar: Calculator
Ribbon: AutoSD I > Miscellaneous



The arc program is for calculating the height, (mid ordinate), arc length, angle, chord, radius and the rise from the chord to the arc at any distance from one end of a circular segment. Select the two values to enter from the list on the right side. Select the Input Unit type. Enter the two values in the format of the unit type and pick Calc. The answers are given in the lower right of the dialog box. If you want a written record in your drawing, pick “Print results to screen” and pick a point.

Arc Length	12’-0		
Angle:	44.9999	Chord:	11'-8 5/16"
Height:	1’-1 15/16"	Radius:	15'-3 5/16"

Pick OK to close the program. The next time the program is opened the previous input and answers are given.

CALC

Pulldown: AutoSD > Calculator > Calc > Decimal
Pulldown: AutoSD > Calculator > Calc > FT.IISS
Toolbar: Calculator
Ribbon: AutoSD I > Miscellaneous

This is a calculator. It's basic functions are add, subtract, multiply, and divide. It uses the RPN, (reverse polish notation), method. You type in a number, press enter, type in another number and then press the function key, either +, -, * or /. To continue adding a chain of dimensions type in the next number and press the function key.

```
Command:      Pick [AutoSD > Calculator or AutoSD I > Miscellaneous] > FT.IISS > Calc
FT.IISS CALCULATOR Press SPACE bar or Esc to exit
10.0208 Enter
5.0304 +                RESULT     10.0208 + 5.0304 = 15.0512
15.0512
```

```
6.03 Enter
20.11 +                    RESULT      6.0300 + 20.1100 = 27.0200
27.0200
6 +                        RESULT      6.0300 + 20.1100 = 27.0200
27.0200 x 6.0000 = 163.0000
163.0000
5 +                        RESULT      163.0000 + 5.0000 = 168.0000
168.0000                   Press SPACE bar to exit

Command:
```

For those of you not accustomed to the RPN style and would prefer the standard entry format there is another version of the calculator in the BONUS folder. The file name is CALC.LSP. If you like a calculator that works the same way you would write a math problem out by hand (If you want to add 2+2, you type in 2+2 Enter) you can copy this file to your AUTOSD folder using Windows Explorer.

Other functions of the calculator include 1/X, CHS, X<>Y, SQUARE ROOT and EXPONENT

- 1/X

This is the reciprocal of x. To take the reciprocal simply press the X key, either upper or lower case.

- CHS

This is the change sign function. Press the letter C to change signs.

- X<>Y

This function swaps "X" with "Y". Press the “<” or “,” key. X<>Y will swap two numbers on the screen. If only one number is on the screen then the number on display will be swapped with the number in the "X" register.

The CALC calculator (RPN type only) has three "stack" registers called "X", "Y" and "Z" and a fourth register which is the display. Stack registers enable you to solve complex equations without actually rewriting them. For example, to solve the equation (5 + 3) + (6 x 2) - (15 - 1), use the following keystrokes:

5 Enter 3 + 6 Enter 2 * 15 Enter 1 - - +

"5 Enter 3 +" results in 8 in the display.
"6 Enter 2 *" results in 12 in the display and 8 in the "X" register.
"15 Enter 1 -" results in 14 in the display, 12 in the "X" register, 8 in the "Y" register and 8 in the "Z" register.

"-" subtracts the display (14) from the "X" register (12) resulting in -2 and moves the number in the "Y" register (8) down to the "X" register.

"+" repeats the same process. It adds the display (-2) to the "X" register (8) resulting in the final answer of 6.

The same equation could have been input using only two storage register as:

5 Enter 3 + 6 Enter 2 * + 15 Enter 1 - -

Every time you press Enter you move the numbers up one register and when you press +, -, * or / you move the numbers down one register.

- SQUARE ROOT

To take the square root of a number, press the letter N (upper or lower case). The square root functions will only work in the decimal calculator.

- EXPONENT

To raise a number to a power, type the exponent and then press the letter M (upper or lower case). For example, to solve for 6 raised to the power of 3 follow these key strokes: 6 Enter 3 M. The exponent function only works in the decimal calculator.

You can switch over to the "RIGHT TRIANGLES" program while you are in the calculator by pressing the letter S (upper or lower case). After getting your information, it will switch back to where you left off in the calculator.

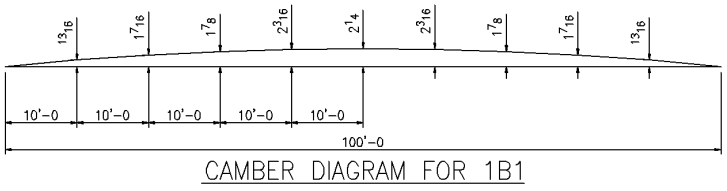
CAMBER

Pulldown: AutoSD > Camber
Ribbon: AutoSD I > Utilities

The CAMBER program is used to find the camber in a beam or truss at any given point and for drawing a camber diagram. After picking CAMBER you are prompted to enter the truss span, (or beam length). A slide drawing is displayed for your reference. After entering the truss span you are prompted to enter the amount of camber at mid span. Next, the program asks for the "Number of panel points from end to center (including end)" and then the "Distance from end to first panel point:" Enter the number of panel points and then the distance from the end to the first point where you want the camber given. The answer will be given in the FT.IISS format and you will be prompted for the distance to the next panel point. This distance should be input as the distance from the previous point to the new point (length of panel of truss). Continue entering the panel spaces until all points are given.

```
Command:
(FT.IISS) Truss span: 100
```

```
(FT.IISS) Camber at mid span: .0204
Number of panel points from end to center (including end): 6
(FT.IISS) Distance from end to first panel point: 10
Length = 100.0000,       A = 0.0013
(FT.IISS) Distance to next panel point <10'-0">: Enter
B = 0.0107
(FT.IISS) Distance to next panel point <10'-0">: Enter
C = 0.0114
(FT.IISS) Distance to next panel point <10'-0">: Enter
D = 0.0203
(FT.IISS) Distance to next panel point <10'-0">: Enter
E = 0.0204
Do you want a camber diagram (Y/N) <Y>:  Enter
Pick point for left end:
Enter Shipping Mark associated with this camber diagram. 1B1
```



CAMBER DIAGRAM FOR 1B1

DECIMAL to FEET

FEET to DECIMAL

Pulldown: AutoSD > Calculator > Dec to Ft
Pulldown: AutoSD > Calculator > Ft to Dec > FT.IISS
Pulldown: AutoSD > Calculator > Ft to Dec > FT-II*SS
Toolbar: Calculator
Ribbon: AutoSD I > Miscellaneous

These two programs are conversion programs. The first one converts decimals of a foot to the FT.IISS format while the second one converts FT.IISS to decimals of a foot.

EXTEND WEIGHT

Pulldown: AutoSD > Calculator > Extend Weight (All)
Pulldown: AutoSD > Calculator > Extend Weight (One line)
Toolbar: Calculator
Ribbon: AutoSD I > Miscellaneous

This command will calculate the weight of an item in the shop bill just by picking the text, (or attribute), for the quantity, description, feet and inches. If you have the shop bill configured with the location of inches as 0 where the length is written as a single text item you will be asked to select the length instead of feet and inches.

If you do not have a configuration selected for the shop bill and this is the first time you use this program in the current drawing session you will be asked to pick a point for the right side of the weight column. Pick any point where the vertical cross hairs of the cursor will be the right end of the text for the weight. If you need to change this point choose the "Locate Weight Column" command directly under "EXTEND WEIGHT" in the pulldown menu. The weight will be written on the same line as the description.

See chapter 10 for information on valid material descriptions.

LOCATE WEIGHT COLUMN

Pulldown: AutoSD > Calculator > Locate Weight Column
Toolbar: Calculator
Ribbon: AutoSD I > Miscellaneous

This command will change the location of the weight column by picking a point. The location is not saved. It is valid only during the current drawing secession. Useful if you do not have the shop bill configured to extend weights but you want to extend some weights afterwards.

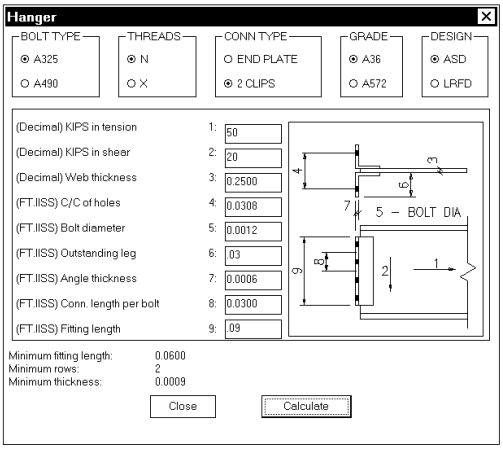
HANGER

Pulldown: AutoSD > Hanger
Ribbon: AutoSD I > Utilities

The hanger program is for calculating the thickness of a connection angle or end plate when the connection has a tension and or shear load.

```
Command:
```

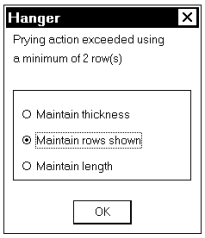
Pick [AutoSD or AutoSD I > Utilities] > Hanger



Make the selections for bolt type, if threads are included in shear plane (N) or excluded (X), connection type, grade of steel used for the connection material and design method.

Enter values in each edit box for your specific connection. If there is no shear load you can enter a value of 0 or leave the box blank. All other boxes must have a value.

After entering all values pick the calculate button.



If prying action is exceeded a dialog box pops up to select how to modify the output.

Option 1 will keep the thickness the same and increase the number of bolts.
Option 2 will keep the number of bolts and increase the thickness.
Option 3 will keep the length and increase the number of bolts and thickness.
Picking the calculate button again will recalculate and let you choose another option if

prying action is exceeded.

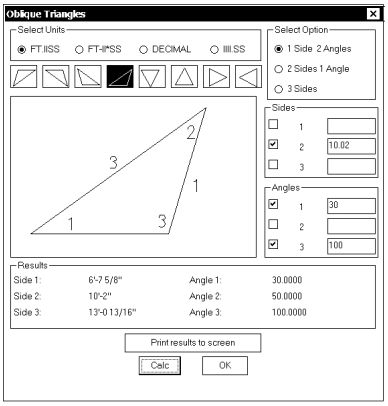
LOADS

Pulldown: AutoSD > Loads
Ribbon: AutoSD I > Utilities

This command is for getting the loads on a member. You input the member shape, size, un-braced length, percent of uniform load and grade of steel and you will be given the load in compression, the load in tension, maximum web shear and end reaction in kips and area of member. If you are looking for the end reaction you must give the length from center of support to center of support as the un-braced length.
If you put a check in the box for Conventional single-plate connection, it will develop a connection using the tables from the AISC 13th edition manual. The shear bolt value has no affect in determining the number of bolts in this case.

OBLIQUE TRIANGLES

Pulldown: AutoSD > Calculator > Oblique Triangles
Toolbar: Calculator
Ribbon: AutoSD I > Miscellaneous



The oblique triangle program will solve any triangle given one side and two angles, two sides and one angle or three sides. It uses icon menus and slides for user information. Select the Units. Changing the units will change the format of the input and results. Select the Option, either 1 Side 2 Angles, 2 Sides 1 Angle or 3 Sides. Under Sides, select which side you want to enter by putting a check in the box and uncheck the others. Do the same for Angles. Enter values for the options selected and pick Calc. If you want to save the results in your drawing, pick "Print results to screen" and pick an insertion point for the text. Press Enter to run the program again.

RIGHT TRIANGLES

Pulldown: AutoSD > Calculator > Right Triangles > Decimal
Pulldown: AutoSD > Calculator > Right Triangles > FT.IISS
Pulldown: AutoSD > Calculator > Right Triangles > FT-II*SS
Toolbar: Calculator
Ribbon: AutoSD I > Miscellaneous

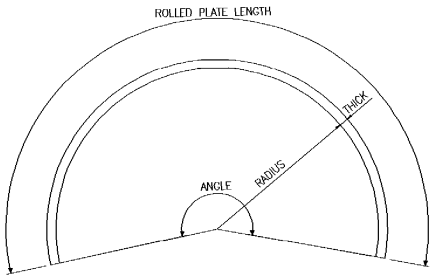
This program is used to solve right triangles. The input can be either one side and the acute angle or two sides. To input two sides you must input the angle as 0. To activate the program you may either pick RIGHT TRIANGLES or type the letter "S" at the keyboard and press Enter.

```
Command:          Pick [AutoSD > Calculator or AutoSD I > Miscellaneous] > FT.IISS > Right
Triangles
Input numbers as FT.IISS
Angle <33.1604>:                          From previous run
Base:
Rise:
Slope: 25.0510
BASE: 21'-3 7/8"          RISE: 13'-11 3/16"          SLOPE: 25'-5 5/8
BEVEL: 7 13/16"           ANGLE: 33.1604
Command:
Input numbers as FT.IISS
Angle <33.1604>: 0
Base:
Rise: 13.1103
Slope: 25.0510
BASE: 21'-3 7/8"        RISE: 13'-11 3/16"          SLOPE: 25'-5 5/8
BEVEL: 7 13/16"         ANGLE: 33.1638
Command:
```

You may enter a different angle at the "Angle:" prompt or use the default angle by pressing Enter. The BASE:, RISE:, and SLOPE: prompts will continue to toggle as long as you press Enter without entering a number.

ROLLED PLATE LENGTH

Pulldown: AutoSD > Calculator > Rolled Plate Length > Decimal
Pulldown: AutoSD > Calculator > Rolled Plate Length > FT.IISS
Pulldown: AutoSD > Calculator > Rolled Plate Length > FT-II*SS
Toolbar: Calculator
Ribbon: AutoSD I > Miscellaneous



Enter the inside radius, thickness and included angle and the program will return the length of plate required.

SECTIONAL PROPERTIES

Pulldown: AutoSD > Sectional Properties
Ribbon: AutoSD I > Utilities

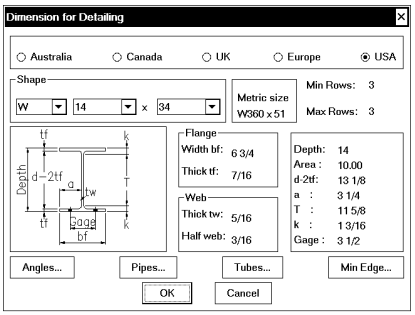
This is a data table program that is used for reference. It will list the dimensional properties of the following shapes:

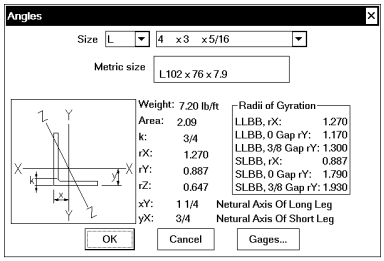
US - C, MC, MT, ST, WT, TS, PS, HSS, HP, M, S, W, CF, L, Turnbuckles, Clevis, Sleeve Nuts
Canada - C, MC, WWT, MT, ST, WT, TS, HSS, DN, HP, M, S, W, WWF, WRF, CF, L
UK - C, BT, CT, RHS, SHS, CHS, ERW, L, J, UB, UC, UBP
Europe - UAP, UPN, HSS, L, HD, HE, HL, HP, HX, IPE, IPN, IPEA, IPEO, IPER, IPEV, DIL, U
Australia - PFC, TFC, BT, CT, RHS, CHS, WB, WC, UB, UC, TFB, EA, UA

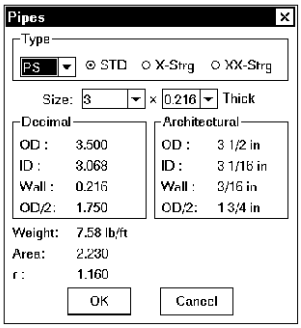
The shape files are located in the "DATA" sub folder of your AutoSD folder.
Canada, UK, Europe and Australia are available only if the drawing is setup for metric.

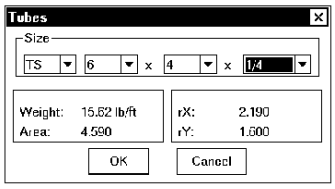
This program uses dialog boxes for selecting shapes and displaying the dimensions. If you pick OK, the size shape selected will be saved and recalled the next time the program is selected from the menu in the current drawing session. If you pick CANCEL, the size shape that was last saved will be recalled the next time.

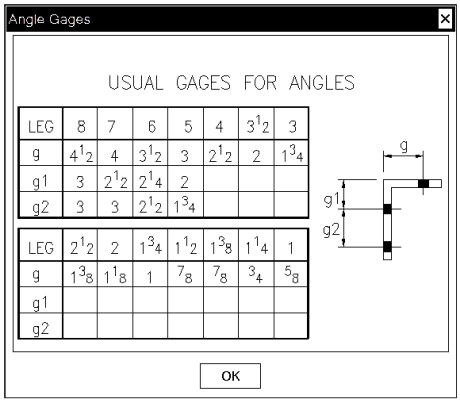
`Command:` Pick [AutoSD or AutoSD II > Beams] > Sectional Properties

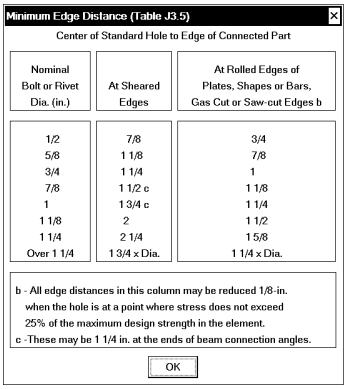


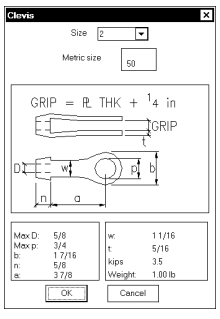


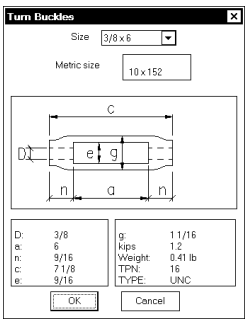


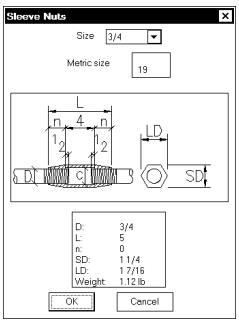












SELECT DIM TO ADD

Pulldown: AutoSD > Calculator > Select Dim to Add > Decimal
Pulldown: AutoSD > Calculator > Select Dim to Add > FT.IISS
Pulldown: AutoSD > Calculator > Select Dim to Add > FT-II*SS
Toolbar: Calculator
Ribbon: AutoSD I > Miscellaneous

This command will calculate the sum of all numbers picked. You may select the numbers one at a time or use crossing or window or a combination of each. If you select a non-legal text string or any non-text entity it will be given a value of 0 and not affect the total.

Using the DECIMAL format of this program is a quick way to get a total of the weight for a completed assembly. You can use "window" or "crossing" to make selections. Non-text items selected will be filtered out. Text strings that contain letters will evaluate to 0. The total is also the default number the next time the CALCULATOR is used.

Valid text strings are listed below. All numbers may be positive or negative and may be enclosed in parenthesis.

1'-3 1/4	1'-0	3 1/4	1/4
1'-3 1/4	1'-0	3 1/4	1/4
15.25	12	3.25	.25

There is one command under DECIMAL and one under FT.IISS. After selecting and adding dimensions the total is shown on the command line in the format for the one selected. The FT.IISS will give the answer in architectural units. If architectural dimensions are added using the DECIMAL format the answer will be in inches. If decimal dimensions are added using the FT.IISS format the answer will be in architectural units with the decimal dimensions representing inches. This can be used to convert a number by selecting only one number to add.

SELECT LINE TO ADD

Pulldown: AutoSD > Calculator > Select Line to Add > Decimal
Pulldown: AutoSD > Calculator > Select Line to Add > Architectural
Toolbar: Calculator
Ribbon: AutoSD I > Miscellaneous

This command will calculate the total length of all lines and polylines picked. You may select the lines one at a time or use crossing or window or a combination of each. The length is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.

TOTAL WEIGHT

Pulldown: AutoSD > Calculator > Total Weight
Toolbar: Calculator
Ribbon: AutoSD I > Miscellaneous

This command will total all of the weights selected and write the total in the shop bill.

Number of risers

Pulldown: AutoSD > Calculator > Number of risers
Ribbon: AutoSD I > Miscellaneous

This command will ask for the height from floor to floor and then a riser height to try. After entering the riser height you are given the number of risers, the actual rise height required and any gain or lose. You are again prompted for the number of risers using the previous number as the default. Press enter to end the command or enter a different number for a new result.

Chapter 6 – Dimensioning

CUSTOM FONT

When you start a new drawing a custom text style is loaded called ASD_BKAP. Another style is available called ASD_FONT. ASD_FONT uses the tilde (~) character while ASD_BKAP uses the backward apostrophe (`) character. The tilde requires holding the shift key down while the backward apostrophe does not. You may use either style. To change styles you must run the program FONT.LSP located in the BONUS folder of AUTOSD.

If you decide to change styles to ASD_FONT at a later time after you have created drawings or you have been using an older version of AUTOSD that did not offer this option you can still change styles. When you open an old drawing created with the ASD_BKAP style and you are now using the ASD_FONT style, use the program ASD_STYLE located in the AutoSD > Modify pulldown menu or the AutoSD I > Utilities ribbon menu to change the existing text in the current drawing.

The following describes the style ASD_BKAP. If you are using the style ASD_FONT simply substitute the ` with a ~. These text styles give you custom fractions, symbols, superscript and subscript text. Type a backward apostrophe (`) or tilde (~) and then the character to type in a custom character. For example, if you want a center line symbol type `C. A fraction is typed as `1 for 1/16, `4 for 1/4 etc. Listed below are the symbols and tilde character.

Symbol	Key	Symbol	Key	Symbol	Key	Symbol	Key
$^{1}/_{16}$	`1	$^{1}/_{8}$	`2	$^{3}/_{16}$	`3	$^{1}/_{4}$	`4
$^{5}/_{16}$	`5	$^{3}/_{8}$	`6	$^{7}/_{16}$	`7	$^{1}/_{2}$	`8
$^{9}/_{16}$	`9	$^{5}/_{8}$	`0	$^{11}/_{16}$	`q or `Q	$^{3}/_{4}$	`w or `W
$^{13}/_{16}$	`e or `E	$^{7}/_{8}$	`r or `R	$^{15}/_{16}$	`t or `T	$^{7}/_{64}$	`y or `Y
(CTRD)	`; or `:	$^{C}/_{C}$	`l or `L	∅	`[ or `{	$\underline{\text{L}}$	`g or `G
$\underline{\text{R}}$	`f or `F	$\underline{\text{R/L}}$	`h or `H	⌋L	`] or `}	℄, CHS	`c or `C
℞	`p or `P	Ø	`o or `O	(AS SHN)	`j or `J	(OPP HND)	`k or `K
(AS SHN AND NTD)	`u or `U	(OPP HND AND NTD)	`i or `I	℞LT	`d or `D		

$^{13}16^{O}$ would be entered as `E`O

Superscripts and subscripts are made by preceding the word or letters with a `S for superscript or `s for subscript. `SSUPERSCRIPT will look like $^{\text{SUPERSCRIPT}}$. `sSUBSCRIPT will look like $_{\text{SUBSCRIPT.}}$ To type 30^{o} you would enter it as 30`So.

Symbol	Key	Symbol	Key	Symbol	Key
Right Tri	S	⊗, 1/x	X	C/C	L
X<>Y	<	$\sqrt{X}$	N	X^{Y}	M
Flip Screen	F2	Coords	F6	Grids	F7
Ortho	F8	Snap	F9	Tablet	F10

A card is provided for your use showing the location of each symbol on the keyboard.

Some of the symbols are for use with the calculator. The locations of these symbols are shown to the left.

There are two different text styles and seven different fonts supplied with AutoSD. The style names are ASD_FONT and ASD_BKAP. The fonts used with ASD_FONT are asdroam, asdromns, asdromn, asdsimpx, asdtxt and cvspro~. The fonts used with ASD_BKAP are asdfrc96, asdram96, asdros96, asdrom96, asdsim96, asdtxt96, asdfrc96 and cvspro`. These are called BIGFONTs. The default style is ASD_BKAP and the default font is asdros96.

These bigfonts are used in association with fonts supplied with AutoCAD. The bigfonts asdroam, asdram96, asdromns, asdros96, asdromn and asdrom96 are used with the AutoCAD font romans. The bigfonts asdsimpx and asdsim96 are used with the AutoCAD font simplex. The bigfonts asdfrc96, asdtxt and asdtxt96 are used with the AutoCAD font txt.

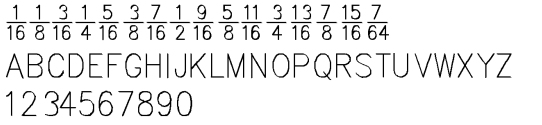
The bigfonts cvspro~ and cvspro` are used with the font cvspro supplied by AutoSD.

To change fonts use the STYLE command.

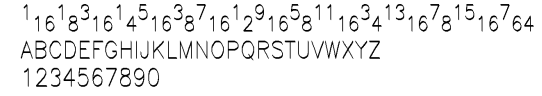
```
Command: STYLE
Text style name (or ?) <ASD_BKAP>:
Existing Style.
Font file <romans,asdros96>: romans,asdsim96        New font. Do not include any spaces.
Big font asdsim96.shx
Height <0'-0">: Enter
Width factor <0.86>: Enter
Obliquing angle <0.0000>: Enter
Backwards? <N>: Enter
Upside-down? <N>: Enter
Vertical? <N>: Enter
ASD_BKAP is now the current text style
```

The different fonts are shown below.

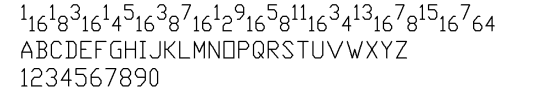
Style ASD_FONT, font: SIMPLEX,ASDSIMPX
Style ASD_BKAP, font: SIMPLEX,ASDSIM96



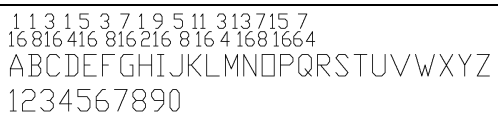
Style ASD_FONT, font: ROMANS,ASDROMNS
Style ASD_BKAP, font: ROMANS,ASDROS96



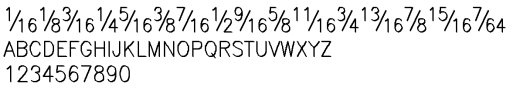
Style ASD_FONT, font: TXT,ASDTXT
Style ASD_BKAP, font: TXT,ASDTXT96



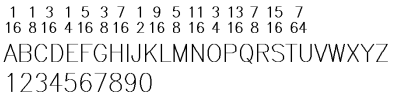
Style ASD_BKAP, font: TXT,ASDFRC96



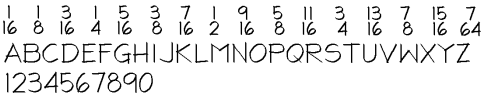
Style ASD_FONT, font: ROMANS,ASDROMN
Style ASD_BKAP, font: ROMANS,ASDROM96



Style ASD_FONT, font: ROMANS,ASDROAM
Style ASD_BKAP, font: ROMANS,ASDRAM96



Style ASD_FONT, font: CVSPRO,CVSPRO~
Style ASD_BKAP, font: CVSPRO,CVSPRO`



DIMENSIONS

DIM. TEXT

Pulldown: AutoSD > Dimension > Dim Text
Toolbar: Dimension I
Ribbon: AutoSD I > Dimension

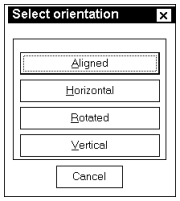
DIM. TEXT is a text command that puts dimensions on without any dimension lines. It operates the same as the TEXT command but uses the FT.IISS format and is for dimensions only.

DIMENSIONING

Pulldown: AutoSD > Dimension
Toolbars: Dimension I, Dimension II
Ribbon: AutoSD I > Dimension

AutoSD uses a custom dimension program. The dimensions will be placed centered between the extension lines and above the dimension line. If the dimension is less than 2 1/2" at a 1"=1' scale it will put tick marks and the dimension text will be placed outside of the extension lines on the side that was picked first, otherwise it will put arrowheads or ticks, per your customer configuration, and the text will be centered. The dimension height will be the same height as the text and will be plotted 1/8" in height no matter what scale you are drawing to. These are pre-set when you run SETUP and are saved with the drawing in a dimension style called AUTOSD. All dimensions are affected by the "MODIFY > SCALE OF DRAWING" command.

```
Command: DIMENSIONS                          Pick from pulldown, toolbar or ribbon menu.
First point or RETURN to select:             Pick a point
Next point:                                  Pick another point.
Enter
Dimension line location:                     Pick a point.
```



A dialog box pops up to select
Select Horizontal.

```
(FT.IISS) Spacing/Text/Dimension <4'-6 11/16">: Enter     If only two points were picked.
Command:
```

If only two points were picked you get the prompt Spacing/Text/Dimension. Typing a T allows you to enter text instead of a dimension. Typing an S allows you to dimension spaces.

If you picked Aligned you would get the prompt "Reference/Enter to use points selected:" Press the enter key and the dimension line will be parallel to the two points picked. Press "R" and enter and you are prompted to select a reference line. The dimension line will be parallel to the line you pick.

If you picked Rotated you would get the prompt "(Decimal) ROTATED dimension line angle <0.0000>:" Enter the angle for the dimension line and press enter

Changes to arrowhead type and size, leader arrowhead, Extension line start and end location and text height are controlled through the Dimension Style dialog box. Colors are controlled through the Customer configuration dialog box. The dimension program uses a custom dimension style. The style name is AUTOSD. For changes to any variables to take affect the changes must be saved to this dimension style.

```
Command: dimstyle
dimension style: AUTOSD
Dimension Style Edit (Save/Restore/STatus/Variables/Apply/?) <Restore>: S
?/Name for new dimension style: AUTOSD
That name is already in use, redefine it? <N> Y
```

Arrowheads

AutoSD uses arrowheads that match AutoCAD's but you can also use your own custom arrowhead. When creating the arrowhead, draw it pointing to the right with the insertion point at the right end. The color for dimension arrowheads must be 5 (blue). If you want to use it for leaders you will need to save it with a different file name and make the color 10. The actual color used is set in the Customer configuration dialog box for colors.

DIMENSION LINE or SINGLE LINEAR DIMENSION

Pulldown: AutoSD > Dimension > Dim Line
Toolbars: Dimension II
Ribbon: AutoSD I > Dimension

This is for dimensions at any angle, horizontal and vertical with no extension lines.

CONTINUE DIMENSION

Pulldown: AutoSD > Dimension > Continuous
Toolbars: Dimension I
Ribbon: AutoSD I > Dimension

This command is for continuing dimensions in a straight line. The dimensions will continue from the last point of the previous dimension in the same direction. Once picked, simply press the enter key to keep continuing.

ARC

ARC DIMENSION

Pulldown: AutoSD > Dimension > Radial > Arc
Toolbars: Dimension I
Ribbon: AutoSD I > Dimension

This command is for dimensioning the length of an arc or an arc distance on a circle. It works the same as the dimension command for straight lines. The arc will be measured in a counter clockwise direction between the two points that you pick.

ARC
CONT

ARC CONTINUE

Pulldown: AutoSD > Dimension > Radial > Arc Continue
Toolbars: Dimension I
Ribbon: AutoSD I > Dimension

This command is for continuing an arc dimension on an arc or on a circle. It works the same as the CONT dimension command for straight lines. The dimensions will continue in a counter clockwise direction.

STATUS, DIMVARS, UNDO, RADIUS, DIAMETER, and ANGULAR are AutoCAD's dimensioning commands. LEADER is a custom leader command that works just like AutoCAD's.

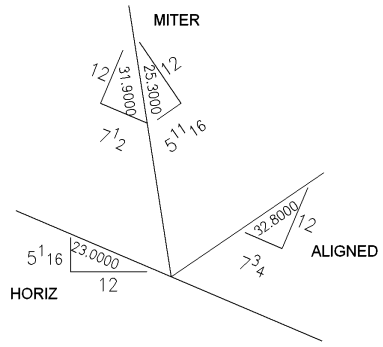
BEVEL

Pulldown: AutoSD > Dimension > Bevel (align)
Pulldown: AutoSD > Dimension > Bevel (horiz)

Pulldown: AutoSD > Dimension > Bevel (miter)
Toolbars: Dimension I
Ribbon: AutoSD I > Dimension

BEVEL (horiz.) calculates the bevel and angle of a line in reference to the horizontal. BEVEL (align) calculates the bevel and angle in reference to another line.
BEVEL (miter) calculates the bevel and angle in reference to two other lines.
Each one places the bevel and angle on the line on the side of your choice. For metric, the bevel base reference is 250.

```
Command: BEVEL, horiz
Command: BEVEL
Select line:                                  Pick a line that is sloping
Select side for symbol:                       Pick one side of the line
Bevel <4 5/8">: Enter
Angle <21.1810>: Enter
Current line-width is 0'-0"                   Symbol is put on line
Command:                                      Finished
```



In response to the "Bevel:" prompt, you may type in a dimension in the FT.IISS format if you don't want to use the default value. You may also type in a different angle at the "Angle:" prompt to be put in the symbol.

CROSSING

Pulldown: AutoSD > Dimension > Crossing
Toolbars: Dimension I
Ribbon: AutoSD I > Dimension

This dimension command is located in the pulldown menu. It allows you to dimension the distance between two or more lines by picking two points. A "rubber band" line is stretched between the two points you pick and all spaces between lines that are crossed by this rubber band line will be dimensioned. The group of lines selected should be parallel.

LEADERS

WORD WRAP

Pulldown: AutoSD > Dimension > Leader (word wrap)

Toolbars: Dimension I, Leader
Ribbon: AutoSD I > Dimension

This leader command can place the text on a single line or wrap it for multiple lines. It can have one leader or multiple leaders

Leader start:
to point:
to point:
Rotation/Leader note: NO PAINT WITHIN 3" OF HOLES
Single Line or Word Wrap? (SL/WW) <SL>: WW
Maximum characters per line <0>: 15
2nd Leader start point (Press [Enter] to exit) Enter

Type an R and press enter to rotate the text to any angle. The default angle is 0 degrees.

QUICK

Pulldown: AutoSD > Dimension > Leader
Toolbars: Dimension I, Leader
Ribbon: AutoSD I > Dimension

This is AutoCAD's multi-line leader command

MULTI LINE

Pulldown: AutoSD > Dimension > Leader (multi line)
Toolbars: Dimension I, Leader
Ribbon: AutoSD I > Dimension

This leader command can place a single line of text or multiple lines. It can have one leader or multiple leaders

Leader start:
to point:
to point:
Rotation/1st Leader note: NO PAINT WITHIN 3" OF HOLES
2nd Leader note (Press X to exit): Enter
2nd Leader start point (Press [Enter] to exit) Enter

Type an R and press enter to rotate the text to any angle.

w/Bubble

Pulldown: AutoSD > Dimension > Leader (w/bubble)
Toolbars: Dimension I, Leader
Ribbon: AutoSD I > Dimension

This leader command can place text in a circle. It can have one leader or multiple leaders

Leader start:

to point:
to point:
1st Leader note (Press X to exit) <>: 1
2nd Leader start point (Press [Enter] to exit) Enter

Dynamic

Pulldown: AutoSD > Dimension > DLeader
Toolbars: Dimension I, Leader
Ribbon: AutoSD I > Dimension

This leader command can place a single line or multiple lines of text while you view the text on screen as it is typed. It can have one leader or multiple leaders

Leader start:
to point:
to point:
Enter text: NO HOLES WITHIN
Enter text: 3” OF HOLES
Enter text:
2nd Leader start point (Press [Enter] to exit) Enter

w/Revision

Pulldown: AutoSD > Dimension > w/revision
Toolbars: Dimension I, Leader
Ribbon: AutoSD I > Dimension

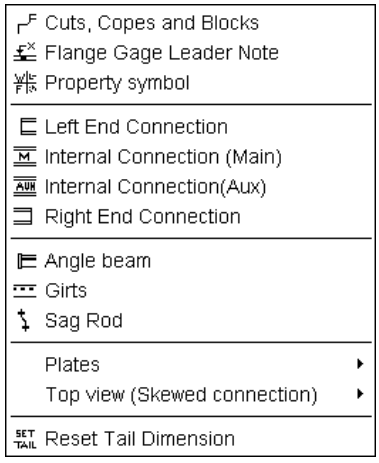
This leader command will place a single line of text with a delta revision symbol under the text. It can have one leader or multiple leaders

Leader start:
to point:
to point:
Leader note: OK
Revision label: 1
Leader start point (Press [Enter] to exit) Enter

Chapter 7 – Beams

All programs in this chapter are under the AutoSD > BEAMS pulldown menu, Beams toolbar and AutoSD II > Beams ribbon menu

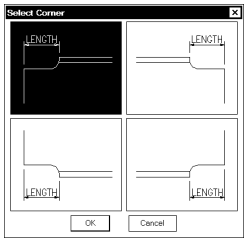
.



Automatically calculate end reaction, rows of bolts, block sizes, flange width cuts, minus dimension, tail dimensions, overall length, cut length .

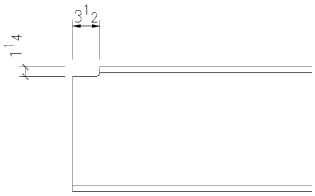
Create configurations for each customer that are automatically loaded when you open a drawing.

Cuts, Copes and Blocks



This program will add a cut to an existing beam detail. The options are Block, Cut & chip or Cut flange width. The way the block is dimensioned is set in the customer configuration for beams.

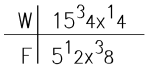
```
Select - Block/Cut & chip/cut flg Width (B/C/W) <B>:
Select top flange of beam:
Select left end of beam:
Pick location of horizontal dimension line:
Pick location of vertical dimension line:
(FT.IISS) Block length: .0308
(FT.IISS) Block depth: .0104
```



Flange Gage Leader Note

This program is for adding the gage to an existing beam. The default will be the gage of the last member selected.

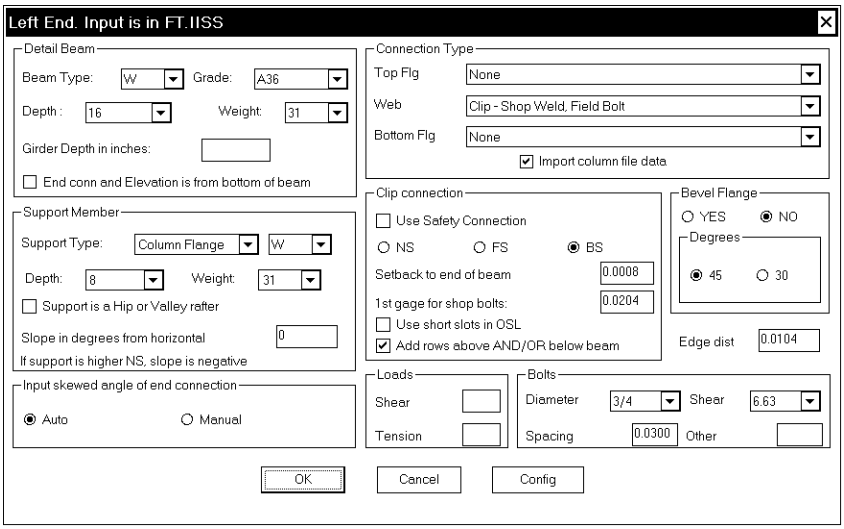
Property symbol



This program will put the symbol, shown to the left, showing the depth and web thickness, (W), and the flange width and thickness, (F) at your insertion point. It uses a dialog box to select the member size.

END CONNECTIONS

Selections for the end connection are made from a dialog box.



Detail Beam

Choose the beam type, grade, depth and weight. If you choose "Girder" for the type, enter the depth in inches or mm for metric.

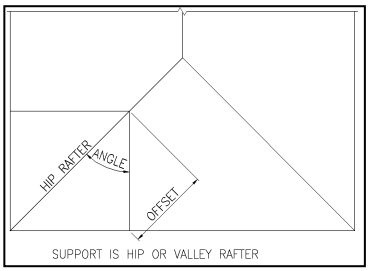
End connection and Elevation are from bottom of beam: Put a check in this box if you want the reference elevation and the connection to be detailed from the bottom of the beam. If the support member is a beam, the reference elevation will also be at the bottom for the support.

Support Member

Next, choose the support type. This can be beam web, column flange, column web, pipe column, tube column, tube beam or user type. User type can be selected if the beam is framing to a masonry wall. If user type is not selected you would then choose the shape, depth and weight. If you chose pipe or tube column the weight selection is disabled. For non-square tube columns, choose the depth that you are framing into. This selection is for determining the "minus" dimension.

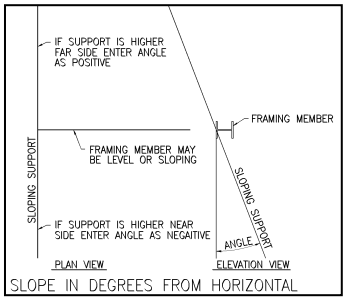
Support is a Hip or Valley rafter

If you are detailing a beam that slopes and is framing into a hip or valley beam you need to check this box. Doing so will adjust the block depths and lengths based on the assumption that the hip or valley rafter and the beam you are detailing are in the same plane. If you are framing into a hip beam the program will also recommend the amount to lower the support so the edge of the flange is in the plane of the beam being detailed. The offset cannot be 0 and the angle cannot be 90.

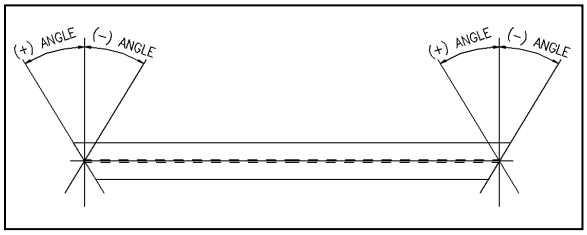


Slope in degrees from horizontal

Normally, the support beam is assumed to be level. If it is sloping you can enter the degrees of slope in order for the program to calculate the block depths correctly. This is different from the Hip or Valley rafter condition above. The support will not be lowered. If the support is higher far side of the beam enter the angle as positive. If it is higher near side, enter the angle as negative. This is important only if the beam being detailed is skewed.



Input skewed angle of end connection



The default setting is *Auto*. This setting will skew the end of the beam automatically based on the offset dimension or angle shown in the plan view on page 102. *Auto* will keep both ends parallel with each other in plan view. If you need one end skewed and the other end square or at some other angle check *Manual* to input the angle.

Positive angles rotate counterclockwise and negative angles rotate clockwise.

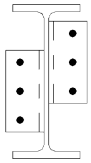
Connection Type

The top and bottom flanges can have nothing, holes, a clip or, if you have purchased the bracing package, you can import a gusset plate that has been previously calculated and saved. The web can have any one of the connections noted under "End Connections" on page **Error! Bookmark not defined.** except for item 9.

Import column file data

If you have created column files you can import the data for certain types of connections to avoid entering it again here.

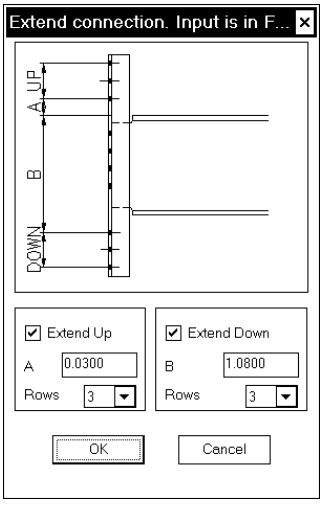
Clip Connection



If a clip is chosen for the web connection you need to select whether it is on the Near Side, Far Side or on Both Sides and the setback from the heel of the clip to the end of the beam. The setback defaults to the setting in the customer configuration but is provided here for a per beam setting. If a shop bolted clip is chosen you can change the "1st gage for shop bolts" dimension. The default is from the customer configuration. This is the distance from the heel of the clip to the first shop bolt in the web of the beam. If the support is a beam, column web or user type and the clip is field bolted you have the option of selecting to "Use Safety Connection". This option will stagger the near side and far side clips. At the left end the far side clip will be dropped down 3" below the near side clip. At the right end the near side clip will be dropped down 3" below the far side clip.

Safety

If the support is a column web, column flange or user type and the connection is a field bolted clip, bent plate or end plate, (not moment), you have the option to "Add row above AND/OR below beam". This option allows you to extend the connection above or below the beam to connect to a gusset plate. The dialog box shown to the right will be displayed to select options and enter number of rows. If you pick "Cancel", the connection will be detailed without being extended.



Bevel Flanges

This is set to "No" by default. If you need the flanges to be beveled for a full penetration weld select yes.

Degrees

If Bevel Flanges is set to Yes you can select the bevel to be 45 or 30 degrees. The default is 45 and will give a clearance of 1/4" from the end of the flanges and 1/2" from the web for wing plate connections. 30 degrees will give a clearance of 3/8" from the end of the flanges and web for wing plate connections.

Loads

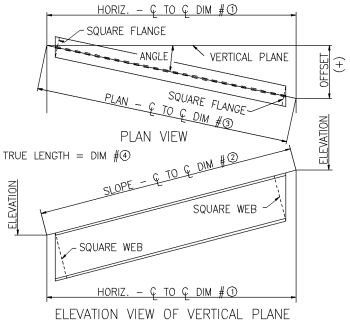
The end reaction will be determined using the beam size, length, grade, design method for country and percent of uniform load from the customer configuration. This load will be compared to the load you enter for the shear load and the greater of the two will be used to calculate the number of bolts in the web connection. The shear load may be left blank if it is not known. The tension load is used to calculate the thickness of the web connection. It may be left blank if there is no tension load.

Bolts

The values in this section are the defaults from the customer configuration but may be changed here on a per beam basis. If you cannot find the shear value you need in the pop down list select "Other" in the list and enter the value in the edit box below the pop down list.

After making all of your selections pick the OK button. At each prompt you may press the enter key to accept the default shown or type in your own dimension.

```
                Main mark: 1B1
Are marks RIGHT and LEFT (Y/N) <N>:
                Quantity: 1
(FT.IISS) Elevation of beam at CL of LEFT end support <0">: Enter
(FT.IISS) Elevation of beam at CL of RIGHT end support <0">: 3
```

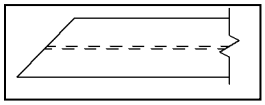


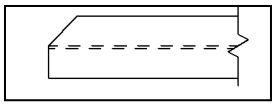
If the beam is not level, the slide picture shown to the left will be displayed. You are asked to select the method of entering control dimensions. The method you select will be used to calculate the true length of the beam and the true lengths between connections. For example: If method #1 is selected as in this case, the slope length, #2, is calculated using the dimension entered as dimension #1 for the base and the difference in elevations for the rise. If an offset is entered and is not 0 the true length, #4, is calculated using dimension #2 for the base and the offset for the rise.

```
Number of dimension to input (1/2/3/4) <1>: Enter
(FT.IISS) Input dimension #1: 15.06
(FT.IISS) Angle/Dimension of Offset of right end in plan, (- is far side) <0>: -2.03
```

Instead of entering an offset dimension you can enter an angle by entering the letter A, press enter and at the angle prompt enter the angle

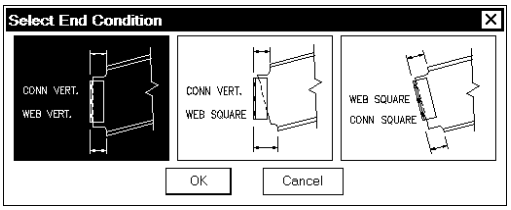
```
Cut flanges square (Yes/No) <N>: Enter
Extend skewed flange cut beyond end of web (Y/N): N
```



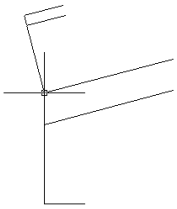


Answering **Y** to extend skewed flange will result in the picture to the far left. Answering **N** will result in cutting the flange off at the web as shown to the left.

Flanges will be extended by default if the "Bevel flanges" option is selected.



For sloping beams, an icon menu is displayed to select the configuration of the end of the beam. As shown in the icon menu to the left there are three choices. (1) The clip and beam end can both be vertical. (2) The clip can be vertical and the beam end can be square. (3) The clip and beam end can both be square. Caution should be exercised when selecting to cut the beam square if the clip is bolted or if the beam has holes in the web.



```
Insertion point:
```

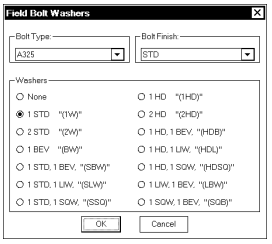
When prompted for the insertion point of the left end an image will be drawn representing the depth and slope angle of the beam. This will enable you to position the beam so that it will not interfere with other details already drawn.

For End Plates without holes enter 0 for number of rows. You MUST pick "YES" for "Plate length = beam depth".

```
End reaction used is 23.44 kips.
Enter number of rows in web <3>: Enter
(FT.IISS) Elevation of top of support beam at connection's vertical work line <0">: Enter
(FT.IISS) Horiz. Minus dimension <-3/16">: Enter
NOTE: Minimum Top flange block depth is 1 1/8".
(FT.IISS) Vert. Distance down to first hole in outstanding leg from W.P. <3">: Enter
Stagger gages of clips (Y/N) <N>: Enter                    This is due to the offset dimension.
(FT.IISS) Hole C/C for web connection <6">: Enter
          Near Side clip...    Since the connection is skewed, the angle size is selected from a dialog box.
NOTE: Leg on web is 4", Outstanding Leg is 3".
Do you want to reverse legs (Y/N) <N>: Enter
Enter angle mark or use <aa>: Enter

          Far Side clip...
NOTE: Leg on web is 4", Outstanding Leg is 3".
Do you want to reverse legs (Y/N) <N>: Enter
Enter angle mark or use <ab>: Enter        Bent clips default to nonstandard marks
```

```
Start shop bill text on LINE <4>: Enter
Number of field bolts in Web (Line 1 of 2) <6>: Enter
(FT.IISS) Total grip less washers <5/8">: Enter
Number of field bolts (Line 2 of 2) <0>: Enter
Shop bill information on mark aa
     Total sub quantity: 1
Shop bill information on mark ab
     Total sub quantity: 1
Add connection to beam file (Y/N) <Y>: Y
```



A dialog box is used to select the bolt type and washers.

The left end is drawn. A top view of the connection and top flange is automatically drawn if the beam is skewed, (offset is not 0). A section showing the bottom flange looking down from the top is automatically drawn if the beam is skewed.

If the support is a beam you will be asked if you want to add the connection to beam file. The default answer is “Y”. Saving the connection will allow you to import the connection as an internal connection when you detail the support member. To take advantage of this connection matching you should start detailing the beams with the least number of internal connections first.

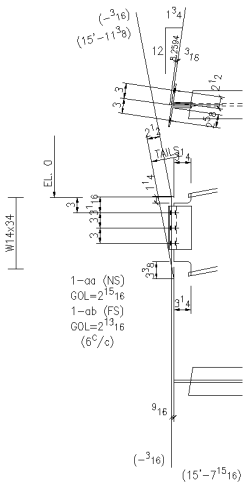
If the support is a column you will be asked if you want to add the connection to a column file.

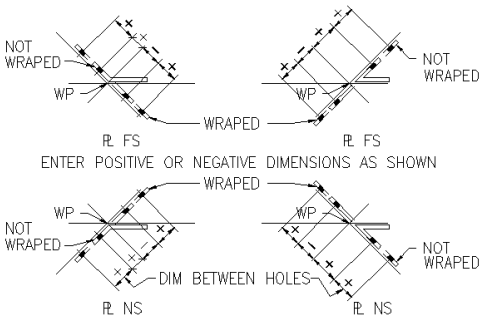
If the connection is a one sided clip or bent plate and the offset is not 0 a slide will be displayed showing the options available for a one sided connection as below to the left. If the dimension from the work point to the first hole is negative the clip will be wrapped.

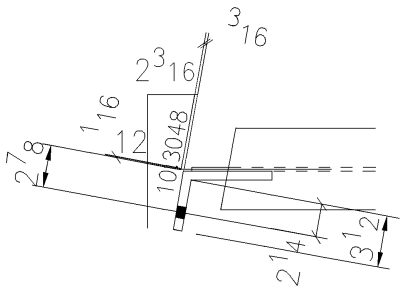
```
The connection is welded on the near side
(FT.IISS) Horiz. WP to hole for web connection <2 7/8">: Enter
(FT.IISS) Horiz space to 2nd column of holes <0">: Enter
(FT.IISS) Minimum edge distance to sides of bent clip <1 1/4">: Enter
        GOL = 2 13/16"
```

The angle size is selected from a dialog box for nonstandard clips.

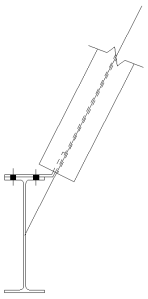
```
NOTE: Leg on web is 6", Outstanding Leg is 4".
Do you want to reverse legs (Y/N) <N>: Enter
```



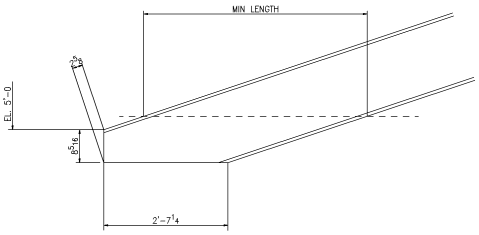




If the angle size selected has unequal legs the long leg, by default, will be shop attached to the beam web. You can reverse this by answering "Y" when prompted to do so.



If you have a skewed beam framing into a column and you want the connection to be to the column flange instead of the web as shown in the sketch to the left you need to select *Column flange* for the support member. This will give you the correct minus dimension and the plate will be bent to fit the flange.

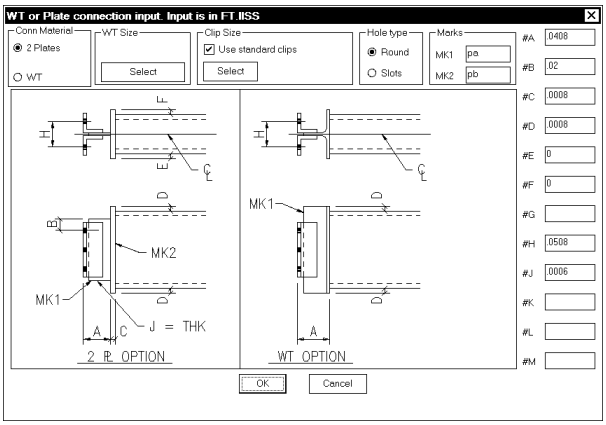


Clipping the lower end of a sloping beam is accomplished by selecting a column, tube beam or user type for the support. Enter "Y" when asked to block the bottom flange. Enter a block length that is greater than the "Min Length" shown in the picture to the left. The program will then ask the following question.

```
(FT.IISS) DEPTH of web cut from top of beam <1 1/8">: .0805
```

This will produce the result shown.

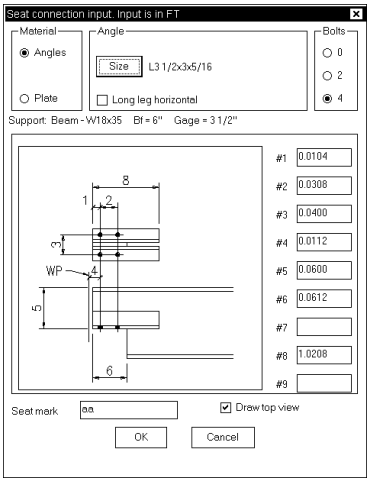
"T" connection



This connection is for tube shapes. The connection can be made from plates or a split "T". It can have two clips or one column or holes.

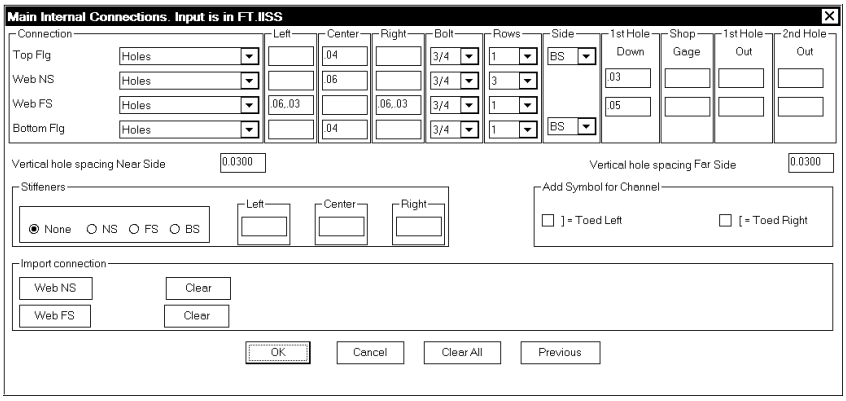
The tube must be level and square with the supports. Skewed and sloping are not options with this connection.

SEAT ANGLE, SEAT PLATE



This connection is for creating a seat using angles or a plate.
The connection may have 0, 2 or 4 bolts.
If there are no bolts then #4 is from the work point to the end of the beam.
You may elect to draw a top view or not.
Do not add "R/L" to the mark if using an angle with holes. It will be added automatically.

M AUX INTERNAL CONNECTIONS (MAIN and AUX.)



The MAIN internal connections are used after the left end of the beam is detailed and before the right end is detailed.

The AUX internal connections are used after the right end of the beam is detailed to add connections after a beam is detailed such as handrail holes or because of a revision. It can be a single connection or you can "chain" them together by pressing enter after the first connection is completed to repeat another connection. If you repeat, you can select the same connection or use a different one. If you select the command from the pulldown menu the connection will not be "chained" to the previous one. You cannot alternate between MAIN and AUX internal connections. You must complete the beam with MAIN before using AUX.

Now let's put some internal connections in the beam. Choose the appropriate box in each section. To recall the last internal connection used pick the Previous button. Pick OK when finished.

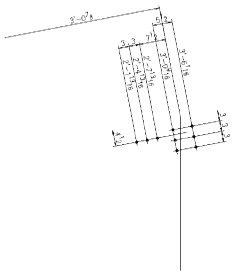
Connections
You can make one selection from each connection. For wing plates, an optional side view section can be drawn. This view is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command. Only the side view will be scaled. If you change the drawing scale before starting the connection you can have the side view drawn larger. This is very

helpful on small beams. Holes placed using Web NS will be in both NS and FS web of tubes. Holes placed using Web FS will be in FS web only of tubes.

Left, Center, Right

After selecting the connection type you should enter a dimension in one or possibly two of these boxes. You can enter a dimension in the *Right* and *Left* boxes or in the *Center* box. You cannot have a dimension in the *Center* box and the *Right* or *Left* box. The program will check for this when you pick the OK button and will display an error message advising you of the problem. Entering a *Left* dimension will draw the connection on the left side of the work line. Entering a *Right* dimension will draw the connection on the right side of the work line. Entering a *Center* dimension will draw the connection on the left and right side of the work line.

Some connections allow you to enter the *Left* and *Right* dimension as a list. Lists are dimensions separated by commas. Example: .0708,.03,.03. Connections which allow lists are holes in flanges, holes in web and stiffener plates. Lists allow you to space holes or stiffeners horizontally. If the dimension is not entered as a list it must be entered using the FT.IISS format if the drawing was setup for architectural units. The example to the left was created using a single *Center* dimension for holes in the near side web with a row of three down 3" and the list noted above for the *Left* dimension for holes in the far side web with a row of one down 4 1/2". This is a typical beam connection for a horizontal brace.



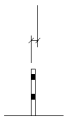
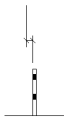
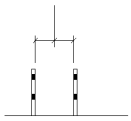
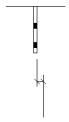
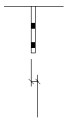
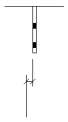
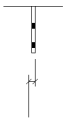
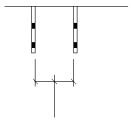
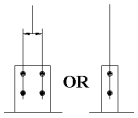
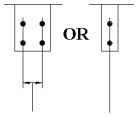
The table below shows the result of entering dimension as a positive and negative. As you can see in the examples, *Left* dimensions move the connection left and *Right* dimensions move the connection right of the work line regardless of being positive or negative. A positive dimension, however, continues to go in the same direction with the connection and a negative dimension reverses the direction of the connection. The exception to this is skewed wing plates. Skewed wing plates are placed on the opposite side of the work line as viewed from the top if the dimension is negative. Entering the angle as negative has no affect. A *Left* dimension rotates the plate left of vertical and a *Right* dimension rotates the plate to the right side of vertical.

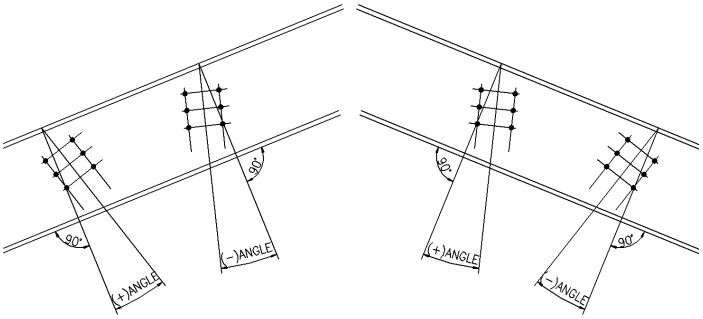
To make a list negative enter the first number as negative. The remaining numbers can be positive. Example: -.04,.03,.03

The examples below show the result of entering dimension as a positive and negative for vertical plates on top and bottom flange. *Left* dimensions move the connection left and *Right* dimensions move the connection right of the work line regardless of being positive or negative. A positive dimension, however, continues to go in the same direction with the connection and a negative dimension reverses the direction of the connection. The front view does not have a *Right* or *Left*. It will always be centered.

Both edge view and front view plates will have either 1 or 2 columns of holes.

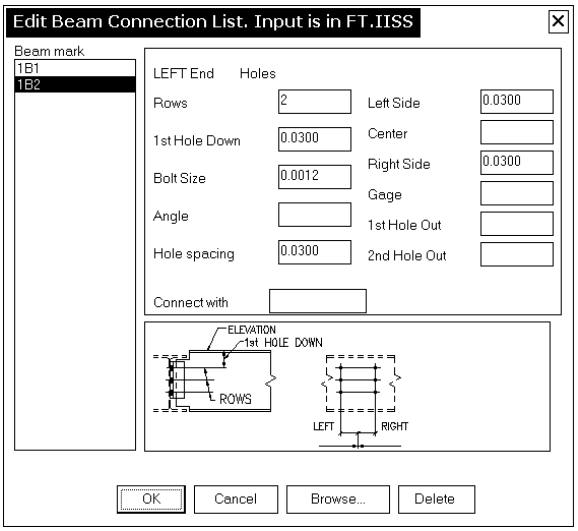
If the beam is sloping, the plates can be either vertical or square with the beam flange.

	Left (+)	Left (-)	Right (+)	Right (-)	Center	Center = 0
Edge View Top Flange						
Edge View Bottom Flange						
Front View Top Flange						
Front View Bottom Flange						



Holes and skewed wing plates may be rotated to any angle in the web of a sloped beam. The angle of rotation is measured from a line that is square with the flange. Positive angles are rotated counterclockwise. Negative angles are rotated clockwise.

Import connection



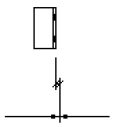
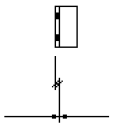
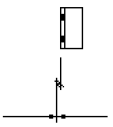
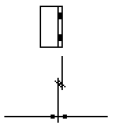
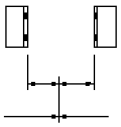
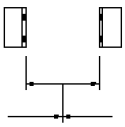
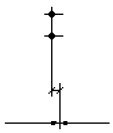
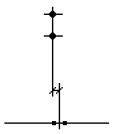
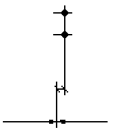
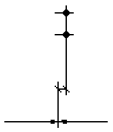
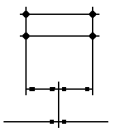
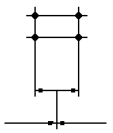
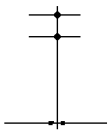
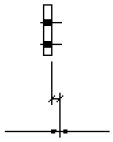
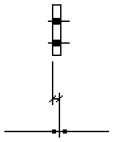
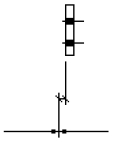
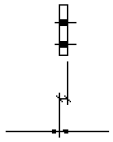
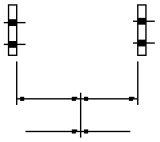
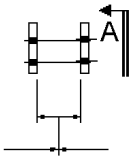
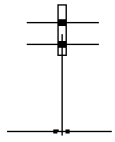
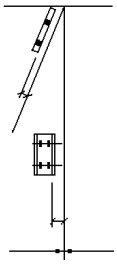
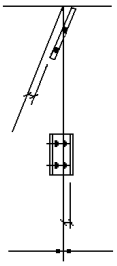
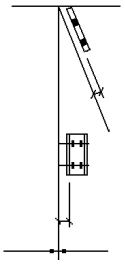
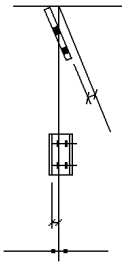
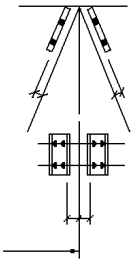
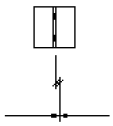
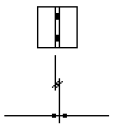
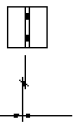
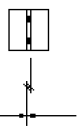
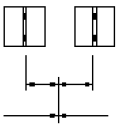
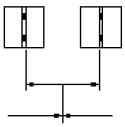
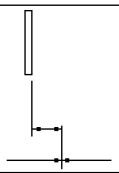
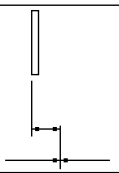
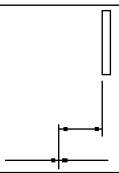
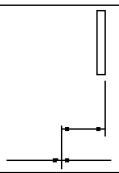
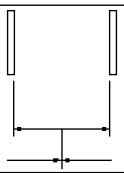
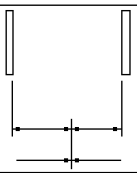
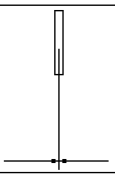
If a web connection is set to *None* in the *Connection* section an import button is enabled. Picking *Web NS* or *Web FS* will open the dialog box shown to the left. This box will list all of the end connections by beam marks that have been saved for the customer you are currently detailing for. To make a selection pick a beam mark and look at the information presented to the right of the list. The first item will be the end of the beam that connection was detailed for. The left end is the piece marked end. The right end would be the other end.

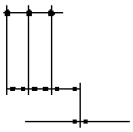
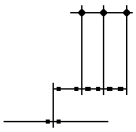
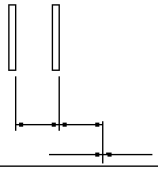
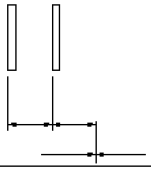
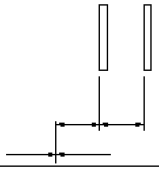
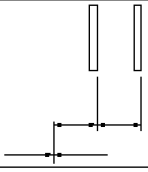
As you detail and save connections, most of the beam marks will be listed twice, one listing for each end. This same box is used to edit connections in the list and can be

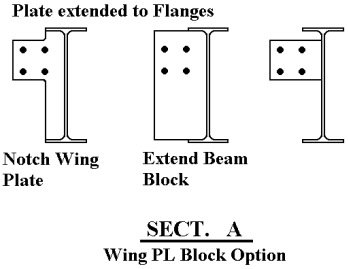
accessed from the beam pulldown menu by selecting *Edit Internal Connection List*. You can edit the dimension by picking on an edit box and changing the dimension. The "Center" dimension is used to put a wing plate or shear lug on the center of the work line for a tube or pipe framing connection. Select *Cancel* to close the box without saving any changes. The default list displayed in the box is from the current customer. Pick the *Browse* button to open bm_conn.lst files that you have saved in other folders. The *Delete* button will delete the currently selected connection from the file. If the correct end is shown pick the OK button.

The plate connections shown below are available from *Internal Connections (Main)***,** *Internal Connections (Aux)* and from *Beams/Plates/Vertical Plate on flange*. They can also be used on top of a column cap plate.

Examples showing the result of entering dimensions as positive and negative for vertical plates located on top and bottom flange or column cap plate.

	Left (+)	Left (-)	Right (+)	Right (-)	Left & Right	Center > 0	Center = 0
Clip Bolted or Welded							Not Available
Holes							
Wing Plate Square							
Wing Plate Skewed						Not Available	Not Available
Split Tee Bolted or Welded							Not Available
Stiff Plate							

	Left (List) (+)	Left (List) (-)	Right (List) (+)	Right (List) (-)
Holes				
Stiff Plate				



SECT. A
Wing PL Block Option

Vertical hole spacing
The defaults are from the customer configuration but may be changed here on a per detail basis.

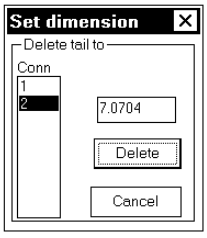
Add Symbol for Channel
Selecting Toed left or Toed right will cause the program to draw the end view of a channel toed in the direction selected and place beside the work line.

After all necessary information has been entered into the Internal Connections dialog box pick the OK button. The insertion point for internal connections is where the vertical cross hair intersects the top flange.

```
(FT.IISS) Center of left support to work line of connection - based on control dim #1: 3
Web connection is (1)Square with flange, (2)Vertical: 1
```

The true length dimension to the work point of the connection from the previous connection is calculated based on the input method selected when the left end was detailed. In this example it is based on control dim #1. If the beam is sloping the connection can be vertical or square with the flanges.

If you insert an internal connection and erase it you must reset the tail dimension. Pick *Reset Tail Dim* from the BEAMS pulldown menu or toolbar and select a number in the *Conn* list box. The numbers represent the connections in the order in which they were detailed. The highest number is the last connection. Selecting a number will display a dimension in the box above the *Delete* button. This is the tail dimension for that connection. Pick *Delete* to reset the tail dimension to the connection number shown above the one selected. This will delete the connection selected and all others below it. Once deleted, they cannot be restored.

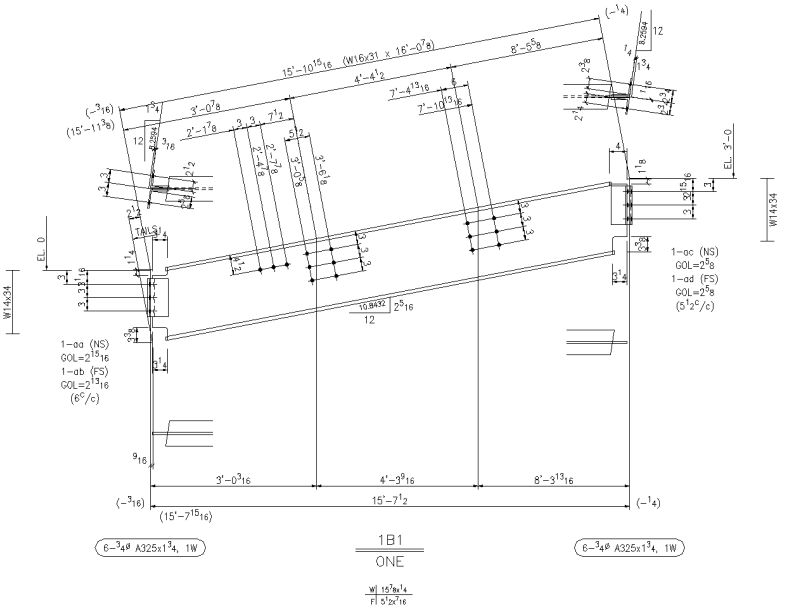


Now we can put the right end on the beam. This uses the same dialog box as the left end but the selections for *Detail Beam* have been disabled. Choose the appropriate box in each section. Choose OK when finished.

```
Cut flanges square (Yes/No) <N>: Enter
Extend skewed flange cut beyond end of web (Yes/No): N
Insertion point:                                    The Insertion point is the
face of the clip.
End reaction used is 23.44 kips.
Enter number of rows in web <3>: Enter
(FT.IISS) Elevation of support beam at connection's vertical work line <3'-0">: Enter
(FT.IISS) Horiz. Minus dimension <-1/4">: Enter
(FT.IISS) Vert. Distance down to first hole in outstanding leg from W.P. <3">: Enter
Stagger gages of clips (Y/N) <N>: Enter
(FT.IISS) Hole C/C for web connection <6">: .0508
          Near Side clip...
```

```
  NOTE: Leg on web is 4", Outstanding Leg is 3".
Do you want to reverse legs (Y/N) <N>: Enter
Enter angle mark or use <ac>: Enter
          Far Side clip...
  NOTE: Leg on web is 4", Outstanding Leg is 3".
Do you want to reverse legs (Y/N) <N>: Enter
Enter angle mark or use <ad>: Enter
Number of field bolts in Web (Line 1 of 2) <6>: Enter
(FT.IISS) Total grip less washers <5/8">: Enter
Number of field bolts (Line 2 of 2) <0>: Enter
Do you want to add ac to the shop bill (Y/N) <Y>: Enter
Do you want to add ad to the shop bill (Y/N) <Y>: Enter
Shop bill information on mark 1B1
Add connection to beam file (Y/N) <Y>: Enter
```

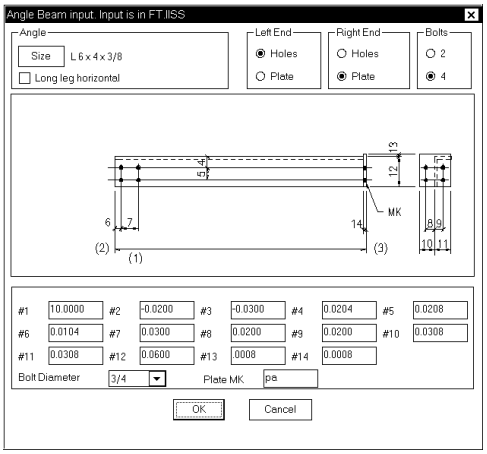
The beam is finished.



NOTE:

- The beam is drawn to actual depth.
- Flange thickness is drawn to 3/4" at a 1" = 1'-0 scale.
- Blocks are drawn to scale.
- Holes are located above (negative) or below (positive) the top flange to scale.
- The elevation will be written at both ends if the beam is sloping.
- The minimum block depth is 1", (25mm).
- You must finish one beam before starting another and before closing the drawing.
- (FT.IISS) Total grip less washers <5/8">: This prompt shows the grip for the bolts in the end connection. It is equal to the clip angle thickness and the web or flange thickness of the support member. If you are framing into a web and you have a beam connecting on the other side, increase the grip by the clip thickness for the other beam. If you are detailing a beam for a wing plate connection the program assumes 3/8" for the wing plate to determine the grip.
- All material is automatically put in the shop bill.
- Inserting an internal connection without selecting a connection will place dimension lines at the insertion point. To be used to add something latter.
- Connections to a beam or column can be saved to a file to be used to detail the beam connection or column.
- Channels must be drawn toed far side only when CNC data is saved.
- The angle entered for skewed wing plates is the angle in the plane that is square with the edge of the plate attached to the web of the beam.

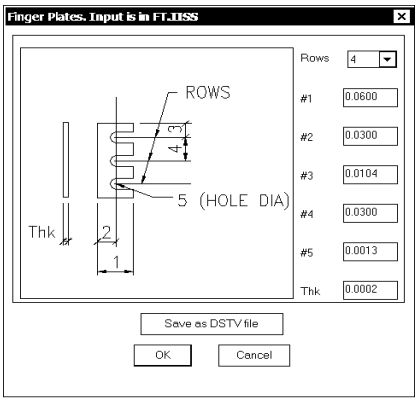
ANGLE BEAM



This draws an angle with either holes or end plates at either end. You can have 2 or 4 holes at both ends.

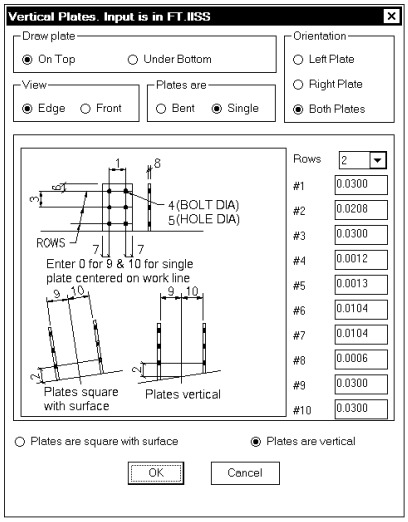
The angle beam will be drawn to the scale set by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.

FINGER SHIM PLATE



This program will draw a detail of a shim plate, place a main mark under the detail and fill out the shop bill or it can write a DSTV file of the plate.. Pick OK to draw the plate and end the program. Pick “Save as DSTV file” to create the file and come back to the input dialog box. The shim plate will be drawn to the scale set by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.

VERTICAL PLATES



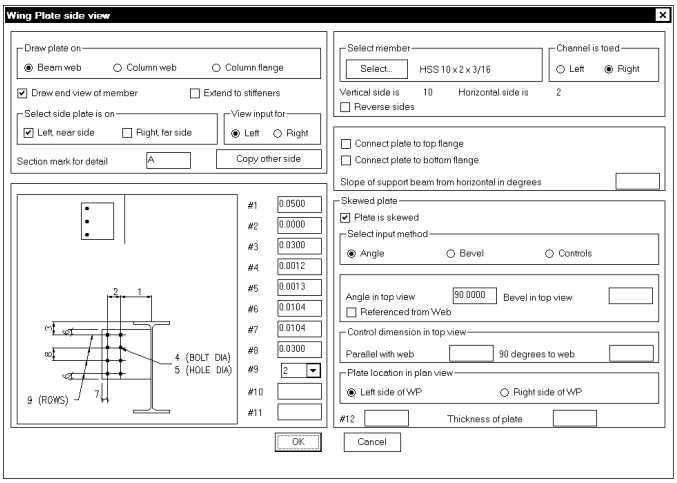
This program will draw one or two vertical plates or one bent plate on top of or below a level or sloping line. You can draw an edge view or a front view. To omit the plate on the left in the edge view, select Single and Right Plate. To omit the plate on the right, select Single and Left Plate. To draw a single plate centered on the work line, Select Single and Both Plates and enter #9 and #10 as 0. To have the plates dimensioned to the outside surface, enter #9 and #10 as a negative. If you change the bolt diameter from the default, the hole size will be noted.

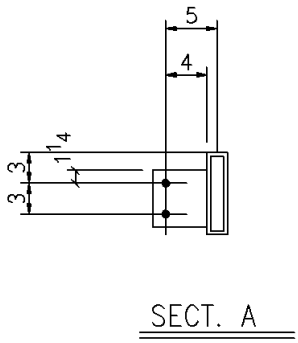
WING PLATE (side view)

Plate Only or Plate and End View of Shape. The wing plate program is for drawing the side view of a shear lug on an existing column detail or end view section. The end view can also be drawn at the same time. The lug can be on either side or both sides and will be dimensioned down from the top. You can have 1 to 14 rows and one or two columns of holes. The plate can be square or skewed with the web of a beam and the beam can be level or sloping. All sections for plates on beams are cut looking to the left.

The wing plate will be drawn to the scale set by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.

Plate & End View of Shape

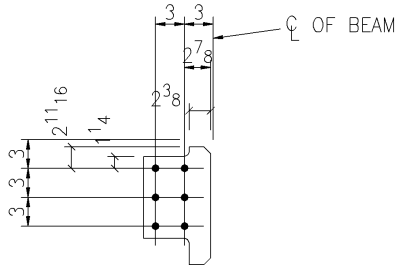





```
Insertion point at top
center of tube:
```

Plate Only

```
Draw plate on Beam web
Left, near side
Member = W14x22
#1 = .03
#2 = .03
#3 = .03
#4 = .0012
#5 = .0013
#6 = .0104
#7 = .0104
#8 = .03
#9 = 3
```

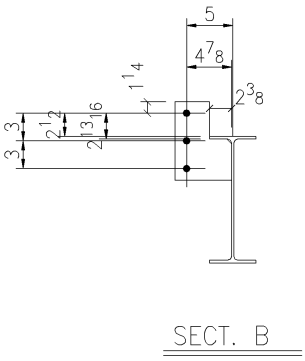



```
Pick center of beam end view on top flange:
```

Entering a positive number for the vertical distance from the W.P. to the first hole will place the first hole below the W.P. A negative distance will place the first hole above the W.P.

The following example uses a negative distance from the top of the beam to the first hole.

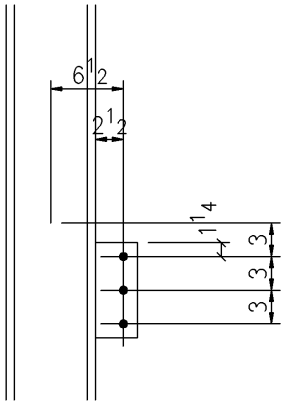
Plate & End View of Shape




```
Draw end view of member
Draw plate on Beam web
Left, near side
Member = W14x22
#1 = .05
#2 = 0
#3 = -.0208
#4 = .0012
#5 = .0013
#6 = .0104
#7 = .0104
#8 = .03
#9 = 3
```

Insertion point at top center of end view:

Plate Only- on an existing W8 x 31 column flange

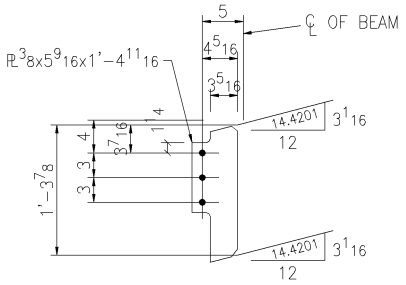



```
Draw plate on Column flange

Right, far side
Member = W8x31
#1 = .0608
#2 = 0
#3 = .03
#4 = .0012
#5 = .0013
#6 = .0104
#7 = .0104
#8 = .03
#9 = 3
```

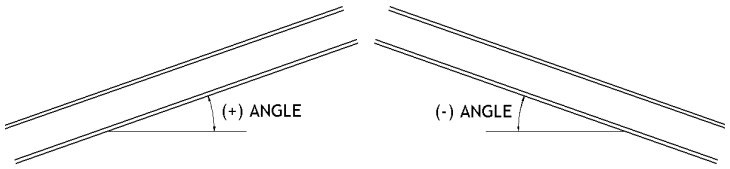
Pick center of column web view at elevation line:

Plate Only - Skewed on Sloping Beam




```
Draw plate on Beam web
Left, near side
Member = W16x26
Connect plate to top flange
Connect plate to bottom flange
Slope of support beam from horizontal = 19.9831
#1 = .05
```

```
#2 = 0
#3 = .04
#4 = .0012                                    Pick center of beam on top flange:
#5 = .0013
#6 = .0104
#7 = .0104
#8 = .03
#9 = 3
#10 = .0002
Thickness of plate = .0006
```



The slope of the beam may be entered as positive or negative as shown above

Note 1: The angle entered can be from the web of the support beam or from a line that is 90^{o} from the web.

Note 2: As you look down on the beam as detailed that the plate is attached to, entering Left would mean the plate is on the left side of a vertical line through the work point. Right would mean the plate is on the right side of a vertical line through the work point.

Note 3: #10 is the half web thickness of the beam framing into the plate.

The end view of the beam will not be drawn if the plate is skewed even if you select to draw the end view.

STIFFENER PLATE (attached)

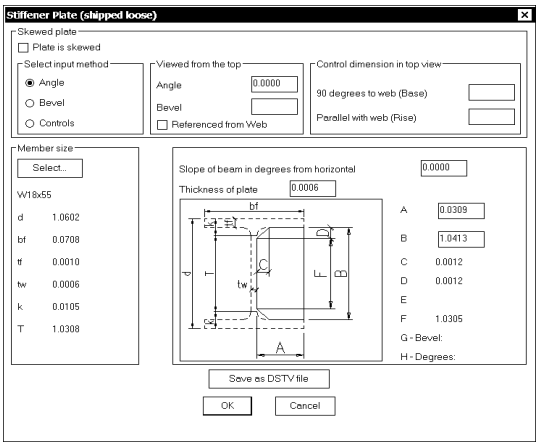
This will draw the edge view of a stiffener plate on an existing detail. It will not put any dimensions.

The stiffener plate will be drawn to the scale set by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.

```
Select a line at one end of stiffener plate:
Select a line at other end of stiffener plate:
Location of center of stiffener plate:
```

The stiffener plate will be drawn square to the second line picked and will be bounded at the ends by the two lines picked.

STIFFENER PLATE (loose)



This will draw a side view detail of a stiffener plate with a main mark or it can write a DSTV file of the plate or do both. Check the box for "Plate is skewed" if the stiffener is not square with the web and enter the angle, bevel or two control dimensions to calculate the angle.

Pick "Select" to select a member size for values d, bf, tf, tw k and T. If the beam is sloped, enter the angle from horizontal. The length of the plate, value "B", will be affected by value #6 under "Shop bolted Clips and Stiff" in your Beam Detail configuration. The width of the plate, value "A", will be affected by the value for "Round Stiffener Plate width to nearest" in your Beam configuration.
Pick OK to draw the plate and end the program. Pick "Save as DSTV file" to create the file and come back to the input dialog box.

The stiffener plate will be drawn to the scale set by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.

SKEWED PLATE (top internal)

The skewed plate program is for drawing the top view of a skewed plate on the web of a beam. The beam must be placed horizontally on the sheet. The plate can be on either side in any direction including 90^{o} to beam, (input angle as 0), and you can have any number of holes. Also, you get to locate where the horizontal, vertical and slope dimensions are to be placed. The skewed plate will be drawn to the scale set by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.

```
Command: Pick [AutoSD > Beams > Plates or AutoSD II > Beams] > SKEWED PL (TOP INTERNAL)
Work point location:                                   Pick a point
Pick quadrant wing plate is in:                        Pick another point
Number of columns: 2
Select to input (Angle/Bevel/Controls) <A>: C
(FT.IISS) Horiz. control dimension: 6.0804
(FT.IISS) Vert. control dimension: 4.1008
Framing into channel (Y/N) <N>: Enter
(FT.IISS) Half web thickness of support beam .........: .0003
(FT.IISS) Offset of plate from work line ..... <3/16">: Enter     May be positive or negative
(FT.IISS) Plate thickness ..................... <3/8">: Enter
(FT.IISS) Slope distance from W.P. to first hole. Min = <3 7/8">: .04
```

(FT.IISS) Distance from first hole to second hole <3">: **Enter**

The plate is drawn

Location of horiz. dimensions: Pick a point

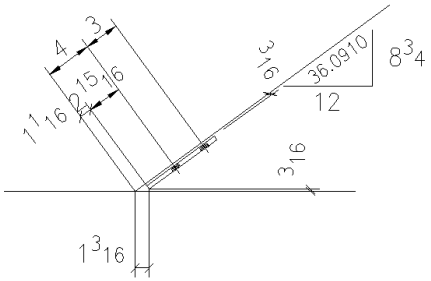
Dimensions are drawn

Location of vert. dimensions: Pick a point

Dimensions are drawn

Location of slope dimensions: Pick a point

Dimensions are drawn



Placing bevel

Select line: Pick a point on the slope

Select side for symbol:

(FT.IISS) Bevel <8 3/4">: **Enter**

(FT.IISS) Angle <36.0910>: **Enter**

Command: Finished

The horizontal and vertical control dimensions may be the actual control dimensions or they can be the bevel. If you select to input the angle or bevel you will be asked if the angle or bevel is in reference to a horizontal or a vertical line.

The picture above is how the plate will be drawn if the offset is positive. If the offset is negative the plate will be on the opposite side of the work line. The offset locates the side of the plate next to the work line in either case.

TOP VIEW (skewed connection)

Right end or Left end.

This program is for drawing the top view of a skewed end connection on the web of a beam. The connection can be on either end or any angle. It uses the same dialog box as the one for detailing the end of a beam so all of the web connection options are available. The first example uses an end plate with the skew angle entered manually. Positive angles rotate counterclockwise and negative angles rotate clockwise.

(FT.IISS) Elevation of beam at CL of LEFT end support <0">: Enter

(FT.IISS) Elevation of beam at CL of RIGHT end support <0">: **Enter**

Enter angle: **36** A slide is displayed to show positive and negative angle rotation.

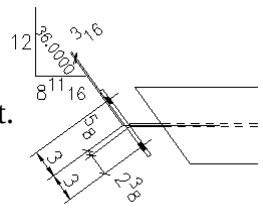
(FT.IISS) End plate thickness: **.0006**

Stagger gage of end plate (Y/N) <Y>: **N**

(FT.IISS) Horiz. Minus dimension <-1/4">: **Enter**

(FT.IISS) Elevation of support beam <0">: **Enter**

Insertion point: Pick a point.

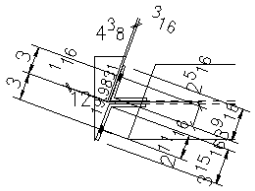


(FT.IISS) Hole C/C in end plate <6">: **Enter**

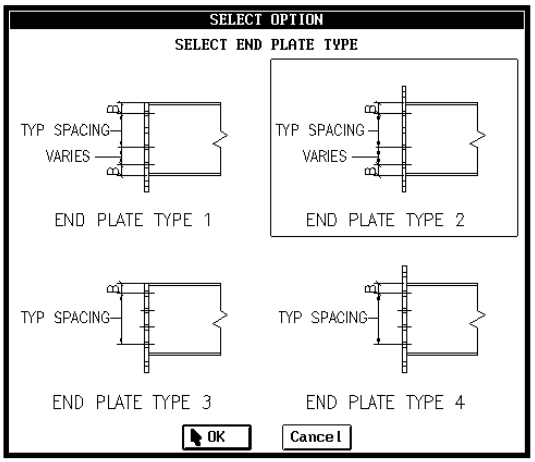
(FT.IISS) Edge distance to sides of end plate <1 1/4">: **Enter**

The following example uses bent plates on both sides of the web with the skew angle entered automatically by entering an offset.

```
(FT.IISS) Elevation of beam at CL of LEFT end support <0">: Enter
(FT.IISS) Elevation of beam at CL of RIGHT end support <0">: Enter
Number of dimension to input (1/3): 1          A slide is displayed to show for user reference
(FT.IISS) Input dimension #1: 11
(FT.IISS) Angle/Dimension of Offset of right end in plan, (- is far side) <0>: -4
(FT.IISS) Thickness bent plate: .0005
(FT.IISS) Horiz. Minus dimension <-3/16">: Enter
(FT.IISS) Elevation of support beam <0">: Enter
Insertion point:  Pick a point.
Stagger gages of clips (Y/N): N
(FT.IISS) Hole C/C for web connection <6">: Enter
(FT.IISS) Edge distance to sides of plate <1 1/4">: Enter
```



MOMENT END PLATE



This program is for drawing an end plate for a moment connection. There are four types to choose from. The moment end plate can be designed and saved in your customer job folder before you detail a beam or it can be designed and saved as it is needed while you detail a beam. The plate can be added to an existing beam detail. One moment end plate design will fit any size beam.

The moment end plate will NOT be drawn to the scale set by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.

Type 1: Holes will be spaced a given distance below the top flange, above and below the bottom flange. The remainder of the plate will be filled with holes at a given spacing.

Type 2: Holes will be spaced a given distance above and below the top flange, above and below the bottom flange. The remainder of the plate will be filled with holes at a given spacing.

Type 3: Holes will be spaced a given distance below the top flange and below the bottom flange. The remainder of the plate will be filled with holes at a given spacing or you can enter the number of rows to use.

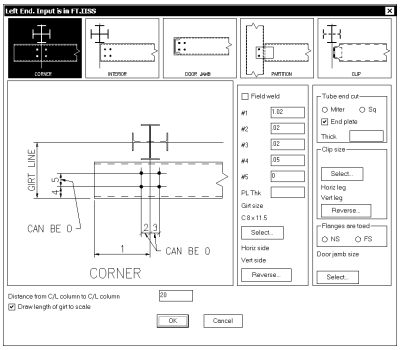
Type 4: Holes will be spaced a given distance above and below the top flange and below the bottom flange. The remainder of the plate will be filled with holes at a given spacing or you can enter the number of rows to use.

```
Command:      Pick [AutoSD > Beams > Plates or AutoSD II > Beams] > Moment End Plate
(FT.IISS) Flange to outside hole ..... "A": .0108
(FT.IISS) Flange to inside hole ..... "B": .0204
(FT.IISS) Spacing of remaining holes .. "C" <3>: Enter
(FT.IISS) Thickness of end plate ..........: .01
(FT.IISS) Edge distance at end of plate ...: .0108
Enter selection: (Save/Quit): S
Customer folder <C:\SSS|95-01\>: Enter
End plate number: 1
Writing output to file ... Done.
```

GIRTS

Pulldown: AutoSD > Beams > Girts
Toolbars: AutoSD > Beams > Girts
Ribbon: AutoSD II > Beams

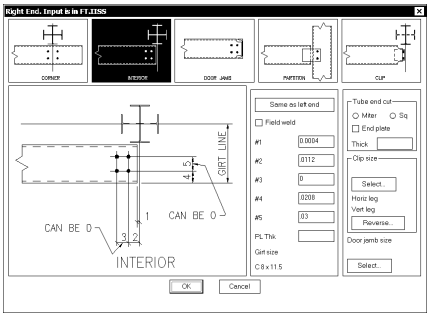
This program is for detailing girts. The girt may have holes, plate or clip in the web for the connection to column, door jamb or another girt or can be field welded. You can have holes for sag rods and door jambs. The ends can be blocked as needed. You must have a customer job folder created and configured for detailing beams before you can detail girts.




```
The last mark was 1B1
      Main mark: 1B2
Are marks RIGHT and LEFT (Y/N) <N>: Y
      Quantity of RIGHTS: 1
      Quantity of LEFTS: 1
```

The dialog box to the left is used to select the left end, enter values for the location of holes, select the girt size, which way the flanges are toed if not saving CNC, enter the length from column to column and select if the length is drawn to scale. Pick OK to enter values for the right end. If the girt is a tube it may be mitered at a corner or cut square with an optional end plate. If the tube does not have equal sides you can select which will be the horizontal side.
Positive dimensions will draw the girt as shown in the slides. Negative dimensions will move holes and the end of the girt in the opposite direction from that shown.

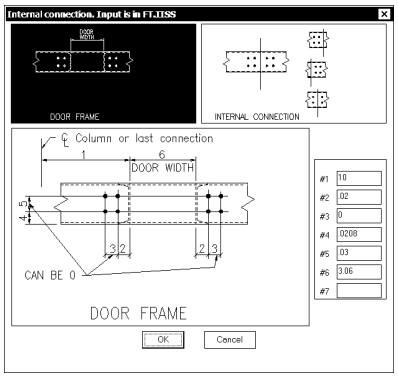
The dialog box to the right is used to select the right end and enter values for the location of holes. Pick OK to continue.



```
How many rows of sag rods: 2
Do you want to enter any SPECIAL block size (Y/N) <N>: Enter
Pick point, top of girt left end:
(FT.IISS) Distance from girt line to C/L of sag rod <4">: Enter
(FT.IISS) Distance from C/L to C/L of sag rods in one group <3">: Enter
```

Can be 0 for one hole.

```
(FT.IISS) Hole size for sag rods (Enter 0 if holes are standard size): 0
Do you want equal spacing for sag rods (Y/N) <Y>: Enter
Do you have any internal connections (Y/N): Y
```



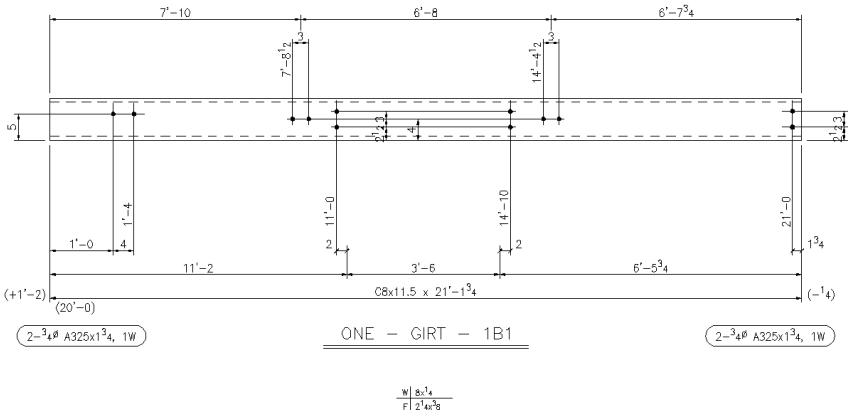
This dialog box is used to select the internal connection and enter values for the location of holes. Pick OK to continue.

```
Pick location for left jamb:          Pick a point
Pick location for right jamb:         Pick a point
```

Pick a "CLEAR" area between holes for vertical dimensions

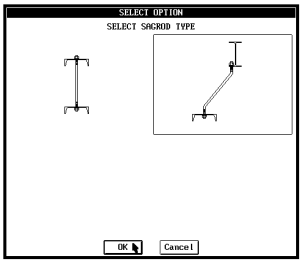
```
Do you have another internal connection (Y/N): N
Place shop bill text on LINE <9>: Enter
```

```
(FT.IISS) Thickness of connection material at left end <5/16>: Enter
(FT.IISS) Thickness of connection material at right end <5/16>: Enter
Command:
```



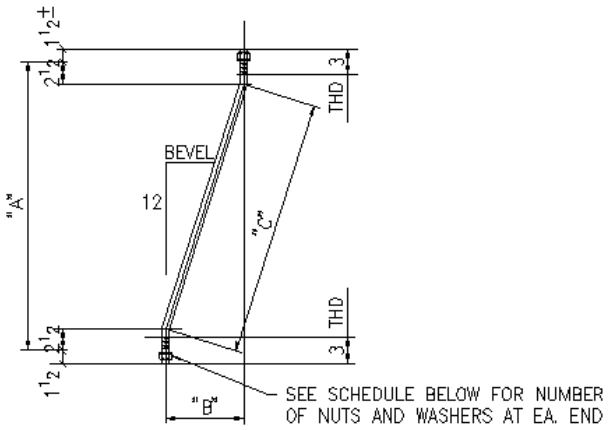
SAG ROD

The SAG ROD program is located in the AutoSD > BEAMS pulldown menu. The sag rod can be straight or bent. Sag rods are put in a schedule, the straight ones in one schedule and the bent ones in another. For this reason, when you are detailing sag rods you should do all of the same type at one time. If you quit the program and come back to detail sag rods later the program will let you add new sag rods to an existing schedule.




```
(FT.IISS) Diameter of sag rod ......: .0008
```

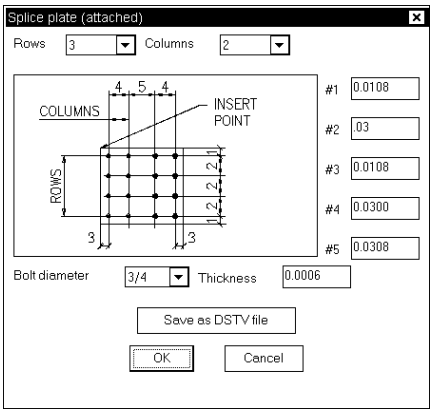
```
Number of nuts at each end .....<1>: Enter
Number of washers at each end ..<1>: Enter
(FT.IISS) Enter dimension ........ A: 5
(FT.IISS) Enter dimension ........ B: 1.02
(FT.IISS) Enter dimension ........ C: .04
(FT.IISS) Enter dimension ........ D: .0112
Insertion point:
                  Move cursor to screen area
      Enter Quantity ...............: 4
      Enter Main ...................: 1R1
Place shop bill text on LINE <11>: Enter
Do you want another sag rod (Y/N) <Y>: N
```



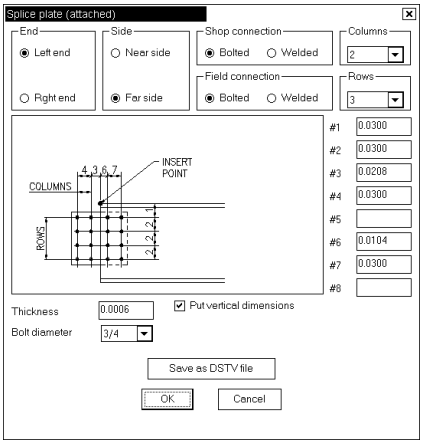
BENT SAG ROD SCHEDULE						
1 NUT(S) AND 1 WASHER(S) REQ'D EACH END						
QTY	MARK	MATERIAL x LENGTH	DIM "A"	DIM "B"	DIM "C"	BEVEL
4	1R1	$1\frac{1}{2}$ø ROD x 5'–$3\frac{5}{8}$	5'–0	$8\frac{1}{4}$	4'–$7\frac{5}{8}$	$1\frac{13}{16}$

SPLICE PLATE (Loose)

This program is for drawing a splice plate with a main mark to be shipped loose or you can write a DSTV file of the plate. You can have any number of rows and columns of holes from 1 to 13. It will put the horizontal dimensions and vertical dimensions. Pick OK to draw the plate and end the program. Pick "Save as DSTV file" to create the file and come back to the input dialog box. The splice plate will be drawn to the scale set by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.




```
Insertion point at top left corner of plate:
      Main mark: 1P1
Are marks RIGHT and LEFT (Y/N) <N>: Enter
      Quantity: 1
Shop bill information on brace 1P1
Place shop bill text on LINE <9>: Enter
```



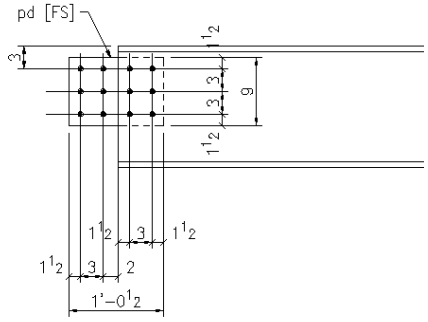
SPLICE PLATE (Attached)

Pulldown: AutoSD > Beams > Plates

This program is for drawing a shop welded or shop bolted splice plate at either end of a beam. You can have from 1 to 13 rows and columns of holes. It will put the horizontal dimensions and you can choose to put the vertical dimensions. This splice plate offers more

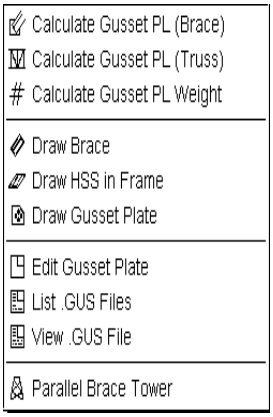
options than the one above but is not affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command. Pick OK to draw the plate and end the program. Pick “Save as DSTV file” to create the file and come back to the input dialog box.

```
Insertion point, top flange at end of beam:
Location of horizontal dimensions:
Location of vertical dimensions:
      Mark for plate
Enter mark or use <pd>: Enter
Add piece pd to shop bill (Y/N) <Y>: N
```



Chapter 8 - Bracing

All programs in this chapter are under the AutoSD > BRACING pulldown menu, Bracing toolbar and AutoSD II > Bracing ribbon menu.



The bracing portion of AutoSD consists of a program to calculate and save gusset plates, a program to edit gusset plate files, a program to view existing gusset plate files, a program to draw gusset plates and a program to draw braces. These programs are supported by icon menus and dialog boxes to select the type of gusset plate and the type and orientation of the brace. A view of the gusset plate type or brace type in the orientation selected is provided for your reference in entering the necessary data for calculations to be made.

Gusset plates should be calculated first. This gives you the information needed to detail the gusset plate and brace. You can calculate all of the gusset plates first on a job and save the information to be used later or you can calculate them as needed.

CALCULATION OF GUSSET PLATES

Determining the number of bolts:
There are four different ways to determine the number of bolts. They are as follows:

1: Load > 0
Area = 0
The number of bolts will be determined by the load.

2: Load = 0
Area > 0
The load will be determined from the area based on the strength of the member. If your brace is double angles you must give the total area of both angles to determine the correct load.

3: Load > 0
Area > 0
The load used will be the greater of the load given or the load as determined in condition number 2.

4: Load = 0
Area = 0
You will be asked to furnish the number of bolts in the brace. The load will be determined by the number of bolts times bolt value times number of shear planes. The number of bolts for the horizontal and vertical vectors will be determined from this load.

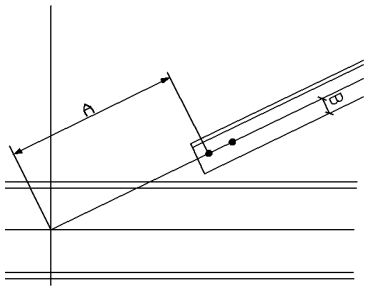
Note!

The tear out of the brace and gusset plate will not be checked if the load and area are input as 0. The number of bolts input will be the number used. The design method used for calculating loads is set in your customer configuration.

Dimension #A

This is the dimension from the work point to the first hole in the brace. If you leave it blank, the dimension will be determined for you based on the rule of thumb clearance method. If you input a dimension, you will be asked to give the desired clearance. If the dimension you input works with the given clearance then it will be used. If it does not work the correct dimension will be calculated based on the given clearance and that will be used. This means that the program can be used to check as well as detail a gusset plate by entering the dimension the detailer used.

If you prefer using a specific clearance rather than the rule of thumb clearance method, simply enter 0 (zero) for "A".



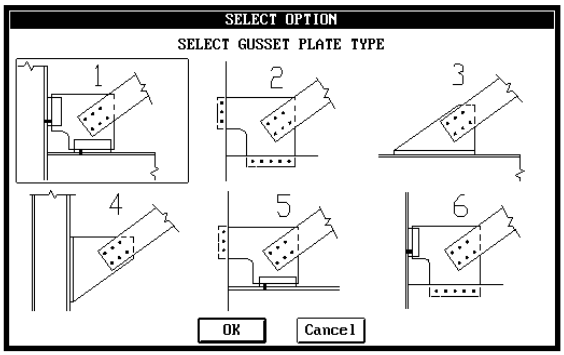
The rule of thumb clearance method:
"A" is based on "B" + 1 1/2" to nearest interference.

Remember, positive dimensions will be in the direction of the picture and negative dimensions will be in the opposite direction.

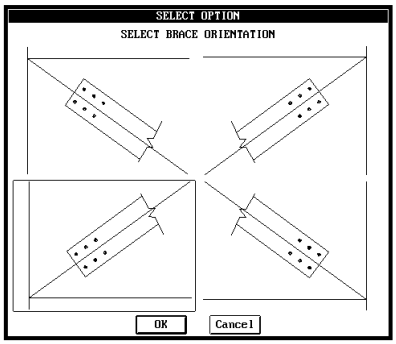
At the end of the program you will be able to get a print of the input and the output, save the output in a file on the hard drive or draw the gusset plate. If you select Draw, the program for drawing the gusset plate will begin. If you select Save, the information will be written to the hard drive in a folder and file name of your choice. A dialog box is used to select the folder. The file name will generally be the connection number. Do not give an extension for the file name. An extension of ".gus" will be added automatically. After saving the gusset plate you can use the GUSSET PLATE DRAW program at any time to draw it. If you do not save it you only have one chance to draw it. It is better to save and then draw.

These files are small in size and several hundred can be saved without using very much room on the hard drive. However, if you want to save the files to a floppy disk you may have to use more than one folder if you have over a hundred files because of the limits of the floppy in being able to store the names of the files.

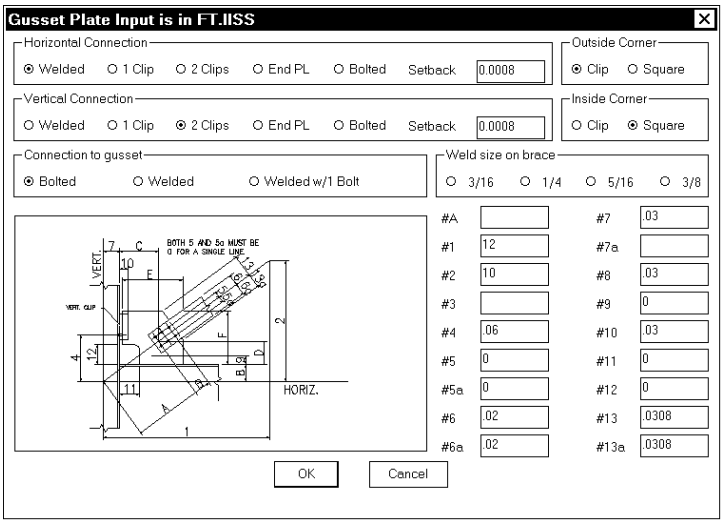
Command: Pick [AutoSD or AutoSD II] > Bracing > Gusset Plate (Brace)



An icon menu appears.
Pick the gusset plate type



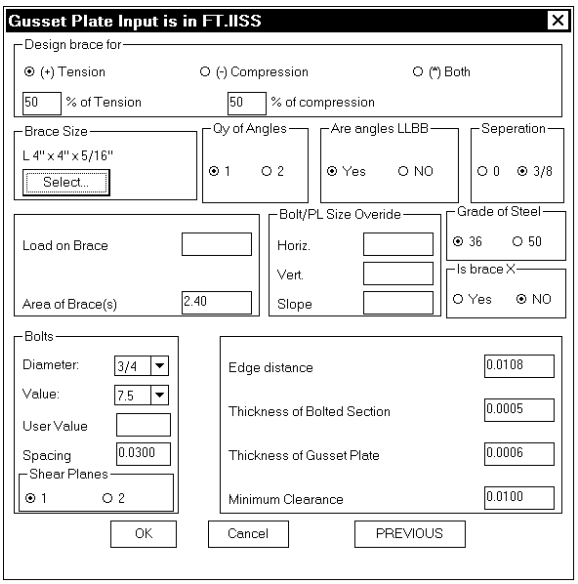
Another icon menu appears for orientation.



After selecting the orientation, a dialog box is put on the screen with the gusset plate type and in the orientation that you picked.

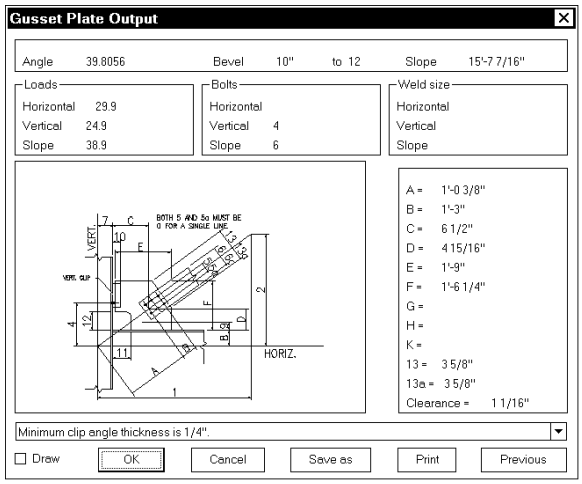
After selecting the options at the top and filling in all of the empty boxes select OK at the bottom of the dialog box.

You can enter 0 for #A if you want to give a specific clearance instead of using the "Rule of thumb" method. You can also input a distance. If it works with the given clearance it will be used. If it does not work the correct distance for the given clearance will be used.



Another dialog box is put on the screen. Select the options at the top and select a brace size. The area of the brace selected will be displayed in the "area" box. If you selected an angle you can select whether you have 1 or 2 angles. If you select 2 angles the value in the "area" box will be doubled and you will need to select the separation.

The three boxes under "Bolt/PL Size Overrides" is for entering values to override the calculated values. If a side of the plate is bolted you would enter the number of bolts to use. If a side is welded you would enter the length of the plate to use. If the program calculates a smaller size than the one given, the size given will be used. Selecting "Other" under bolt value allows you to enter a "User Value" for bolt shear.



If you edit an existing gusset plate file, these boxes will be filled in to force the program to re-calculate the same size plate. If you want to calculate the actual size and number of bolts needed you should delete the values in these three boxes.

When you have entered all the information needed, select OK.

The gusset plate is calculated and the answers are put in another dialog box. You can select to Draw, Save as and Print at the bottom of the dialog box. If you want to make a change you can select previous to go back to the previous dialog box. Picking OK will save the current file and close the dialog box.

If gusset plate tear out had been a factor, a warning message would be printed at the command line. You would then have the option to increase the gusset plate thickness or let the program increase the number of rows until tear out is eliminated.

If the gusset plate is welded to the brace the plate will be sized 1” bigger in the direction of the brace.

Dimensions 5, 6, and 13 are always above the work line and 5a, 6a and 13a are always below the work line. If 5 & 5a, 6 & 6a or 13 & 13a are not the same, that is, your connection is not symmetrical about the work line, you will need to calculate a separate plate for each end of the brace. Gussets calculated in a particular quadrant that are not symmetrical should be drawn only in that quadrant. Otherwise, the plate will not be correct. Plates with holes that are symmetrical about the work line can be drawn in any position.

Note: In order to calculate a gusset plate with a single line of holes offset below the work line you would input 5 as negative and the same absolute value as 5a. In other words if 5a is .01 then 5 must be -.01.

PRINTING FILE

Below is what the print of the input and output looks like. A print like this one without the input or output listed along with a copy of each gusset plate type and each orientation is provided for your use in appendix C. They can be used by other draftsman to write in the input. Your CAD operator can input the information and get a print of the output to be used by the draftsman to detail the gusset by hand. This will be a big time saver for those that have only one CAD station.

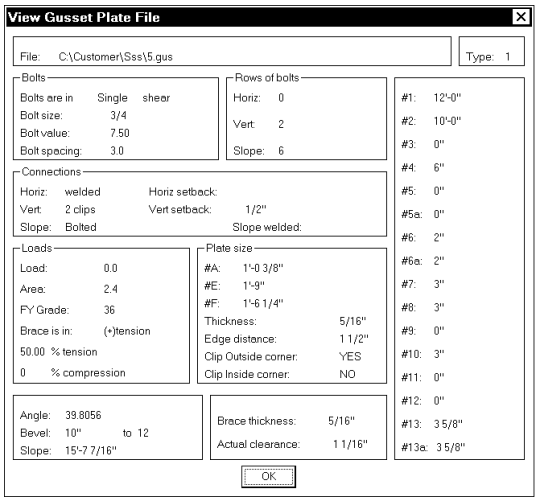
```
Connection NO. 1          TYPE: 1   Quad: 1     DATE: 9-29-2012  10:24am

Load on brace --------------------------: 38.9
Total area of brace(s) -----------------: 2.40
(+)tension (-)comp (*)both -------------: +
    Smaller radius of gyration ---------: 1.240
    Is brace "X" (Yes/No) --------------:
Percent of tensil strength -------------: 50
Percent of compressive strength --------:
Bolt size ------------------------------: 3/4"
Edge distance --------------------------: 1 1/2"
Bolt value single shear ----------------: 7.50
Bolt spacing ---------------------------: 3.0
Grade of steel Fy = --------------------: 36
Grade of gusset plate Fy = -------------: 36
Bolts in (1)Single or (2)Double shear --: 1
Thickness of brace ---------------------: 5/16"
Thickness of gusset plate --------------: 5/16"
Horizontal connection is ---------------: welded
Vertical connection is -----------------: 2 clips     Setback --: 1/2"
Actual clearance is --------------------: 1 1/16"
Corner is clipped
Brace is bolted to gusset plate.

     INPUT                 OUTPUT

#1 : 12.0000    A: 1.0006     Load Slope: 38.9
#2 : 10.0000    B: 1.0300     Load Horiz: 29.9
#3 : 0.0000     C: 0.0608     Load Vert : 24.9
#4 : 0.0600     D: 0.0415     Bolt Slope: 6
#5 : 0.0000     E: 1.0900     Bolt Horiz: 0
#5a: 0.0000     F: 1.0604     Bolt Vert : 4
#6 : 0.0200     G:            Weld Slope:
#6a: 0.0200     H:            Weld Horiz: 3/16" Min. fillet
#7 : 0.0300     K:            Weld Vert : 3/16" Min. fillet
#7a:                          Angle     : 39.8056
#8 : 0.0300                   Bevel     : 10" to 12
#9 : 0.0000                   Slope     : 15.0707
#10: 0.0300
#11: 0.0000
#12: 0.0000
#13: 0.0310
#13a 0.0310

Minimum clip angle thickness is 1/4".
Minimum gusset plate thickness is 5/16"
```



VIEWING FILE

To the right is a view of the file 1.gus that was saved from the calculation of gusset plate type 1. Pick VIEW .GUS FILE from AutoSD > Bracing menu to view a gusset file. Enter the folder name and select the file. This view of the file is provided for reference in case you need to verify an input.

LIST .GUS FILES

This command will give a complete listing of the file names with the extension ".gus" in the folder that you enter. The ".gus" files are created when you save a gusset plate calculation to be used to draw a gusset plate.

EDIT GUSSET PLATE

This command allows you to edit an existing gusset plate file.

DRAWING GUSSET PLATES

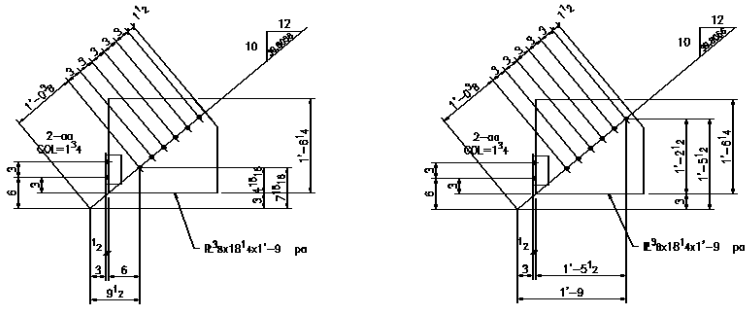
In order to draw a gusset plate, you must first calculate one and save the output in a file (See "CALCULATION OF GUSSET PLATES"). When you select DRAW GUSSET PLATE from the pulldown menu under AutoSD > Bracing you will be asked for the folder and file name of the gusset plate you wish to draw. After giving a valid name you are prompted for the work point location and the quadrant the gusset plate is in. There are 4 quadrants. Upper right, upper left, lower left and lower right. The quadrant you pick does not have to be the same as the one chosen in the calculation program unless the connection is not symmetrical about the work line.

Once a name is accepted, the program starts drawing the plate. After the outline of the plate is drawn and the holes are put in the plate you are prompted for the location of the horizontal dimensions of the plate. These dimensions are from the work point to the first hole for the brace. If the plate has any holes on the horizontal side for a connection of the plate to a beam you are then prompted for the location of the dimensions for these holes. This same procedure is repeated for the dimensions on the vertical side and for the dimensions on the slope.

```
Command:                                    Pick [AutoSD or AutoSD II] > Bracing > Draw > Gusset Plate
Customer folder <D:\ACAD\>: C:\SSS\95-01
Connection number: 1
Reading file ... Done
Work point location:                                                    Pick the work point
Pick quadrant gusset plate is in:                  Upper right, upper left, lower right, lower left.
Are you inserting gusset plate on brace <N>: Enter      If you are, the holes and dimensions
                                                        on the slope will not be drawn they
                                                        will be on the brace.
Piece mark for vertical clip:
Enter angle mark or use <aa>: Enter
Add piece to shop bill (Y/N) <Y>: Enter
Shop bill information on aa
      quantity <2>: Enter
Place shop bill text on LINE <4>:
      Description: L3x3`8x`5
(FT.IISS) Gage of clip: .0109
Horiz. & vert dimensions go to (1)first, or (2)last hole <1>: Enter
Location of horiz. dimensions for brace holes:      Pick location above or below the plate
```

`Location of vertical dimensions for brace holes:`	Pick location to the right or left the plate
`Location of dimensions for vertical holes:`	Pick location to the right or left the plate
`Location of dimensions on slope:`	Pick location above or below the work line
`Location of arrow for plate size:`	Pick one side of the plate
`Location of plate description:`	Pick location
`(FT.IISS) Plate thickness <3/8">:` **Enter**	
`Enter plate mark or use <pa>:` **Enter**	
`quantity:` **1**	
`Place shop bill text on LINE <5>:` **Enter**	
`Placing bevel`	
`Select line:`	Select a point on the work line
`Select side for symbol:`	
`(FT.IISS) Bevel <10">:` **Enter**	
`Angle <39.8056>:` **Enter**	
`Command:`	Finished.

The plate size is rounded up to the nearest 1/8". The detail of the gusset plate is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command. The plate below on the left used the option for horizontal and vertical dimensions to go to the first hole. The plate on the right used the option for horizontal and vertical dimensions to go to the last hole.



If you want to calculate the weight of the plate use the CALCULATE GUSSET PL WEIGHT command and simply pick one edge of the plate and enter the thickness. The weight will be the finished weight with corners clipped and blocked. The scale is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.

DRAWING BRACES

The brace program will draw several types of braces, "X" braces and simple braces. You should be able to detail a sheet of braces in about an hour. This includes calculating the gussets and filling out the shop bill.

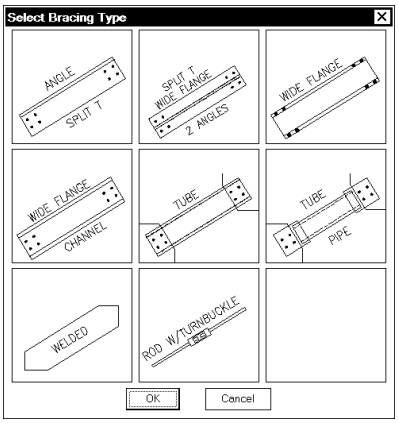
The different types are listed below.

- Angles - Single or double, simple or "X"
- WT - Flange view, simple or "X"
- Wide flange - Web view, simple only. Flange view, simple or "X"
- Channel - Web view, single or double, simple or "X"
- Pipe - Simple or "X"
- Tube - Simple or "X"

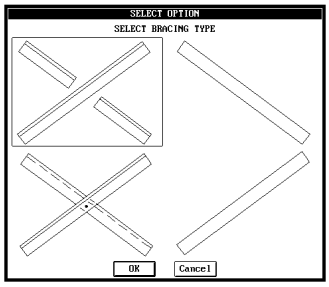
The gusset plate calculation program should be run first to get the necessary input such as number of rows and distance from work point to first hole in brace. The brace program will draw the brace to scale and to the actual bevel. It will calculate tail dimensions, work lengths, cut lengths, the intersection point for "X" braces, and number of fillers for double angle braces. "DRAW" will be taken out of simple single angle braces.

For simple braces, the brace will be drawn in the same direction as the one you pick from the icon menu for orientation. For "X" braces, the braces will be drawn in the same direction as shown in the slide view that is used to get the input data.

`Command:` Pick [AutoSD or AutoSD II] > Bracing > Draw > Brace



An icon menu appears. Pick the material shape.

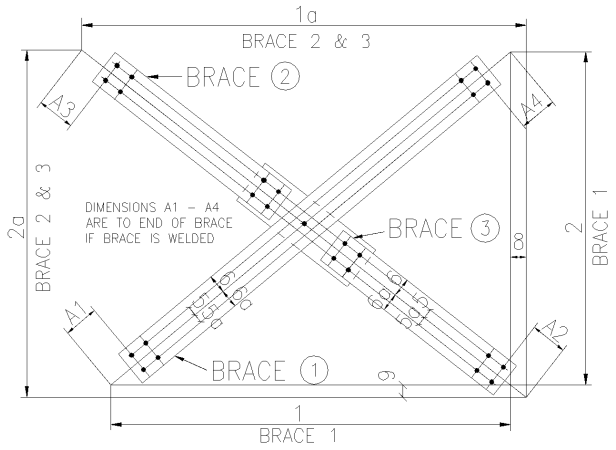


Another icon appears. Pick the type of brace.

```
Customer folder </SSS/95-01>: Enter
Is brace double angles (Y/N): Y
Are fillers bolted (Y/N): Y
Is brace in compression (Y/N): N
Select Brace Size                                   Select a L3 1/2 x 3 x 5/16
(FT.IISS) Separation of angles: .0006
Read connection file for #A1: Y      If you answer "N" you must input the information manually
Read connection file for #A2: Y
Read connection file for #A3: Y
Read connection file for #A4: Y
(FT.IISS) Bolt spacing <3">: Enter
```

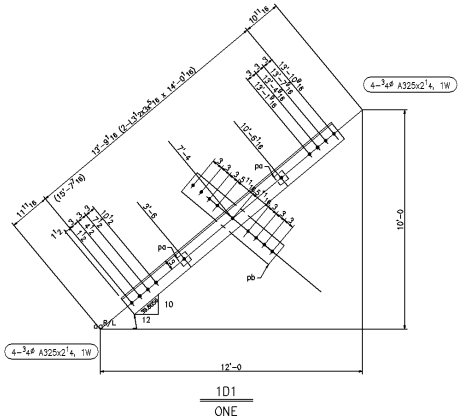
```
Connection file for #A1: 1
Draw length #A1 to scale (Y/N) <Y>: Enter
Connection file for #A2: 1
Draw length #A2 to scale (Y/N) <Y>: Enter
Connection file for #A3: 1
Draw length #A3 to scale (Y/N) <Y>: Enter
Connection file for #A4: 1
Draw length #A4 to scale (Y/N) <Y>: Enter
Bolt size:  (5)5/8"  (6)3/4"  (7)7/8"  (8)1"  (9)1 1/8"  (10)1 1/4" <6>:
(FT.IISS) Input #8 <0">: Enter
(FT.IISS) Input #9 <0">: Enter
Insert point at left end:
```

Pick a point for the lower left end of brace.



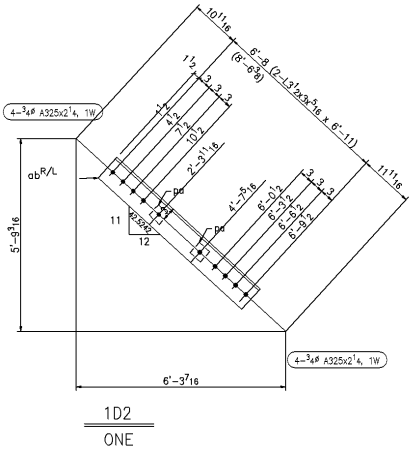
The main brace (shown as Brace 1 in the slide to the left) is drawn first. The insertion point is the lower left end. Brace 1 and 2 will be drawn second and third and their insertion point is the upper left end.

```
The last mark was 1
              Main mark: 1D1
Are marks RIGHT and LEFT (Yes/No) <N>: Enter
               Quantity: 1
Sub assembly mark for brace: aa
Shop bill information on mark 1D1
Place shop bill text on LINE <11>: Enter
Shop bill information on mark aa`H
Sub assembly mark for filler plate: pa
Shop bill information on mark pa
(DECIMAL) Bolt value single shear <6.6300>: 9.9
ASTM = (A36/A572/A588/A992) <A36>: Enter
(FT.IISS) Thickness of center plate <3/8">: Enter
*** For same area as brace, Width of plate should be
11 3/4".***
(FT.IISS)    Width    of    center    plate    using
Load/Thickness/(0.6Fy) = <7">: Enter
Sub assembly mark for center plate: pb
Shop bill information on mark pb
```



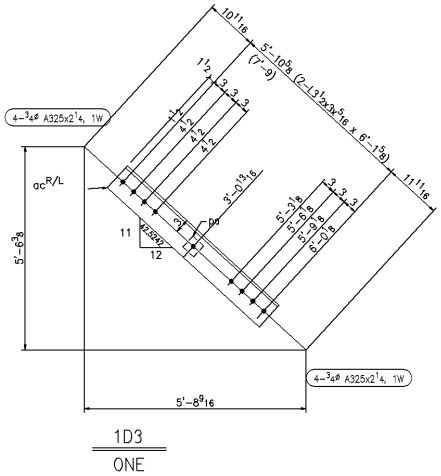
First Brace is drawn

```
Insertion point at left end of brace #2:
```
Pick a point for the upper left end of brace.
```
The last mark was 1D1
              Main mark: 1D2
Are marks RIGHT and LEFT (Yes/No) <N>: Enter
               Quantity: 1
Sub assembly mark for brace: ab
Shop bill information on mark 1D2
Place shop bill text on LINE <18>: Enter
Shop bill information on mark ab`H
Sub assembly mark for filler plate <pa>: Enter
Shop bill information on mark pa
```



Second Brace is drawn

```
Insertion point at left end of brace #3:
```
Pick a point for the upper left end of brace.
```
The last mark was 1D2
              Main mark: 1D3
Are marks RIGHT and LEFT (Yes/No) <N>: Enter
               Quantity: 1
Sub assembly mark for brace: ac
Shop bill information on mark 1D3
Place shop bill text on LINE <24>: Enter
Shop bill information on mark ac`H
Sub assembly mark for filler plate <pa>:
Shop bill information on mark pa
Command:
```



Third Brace is drawn

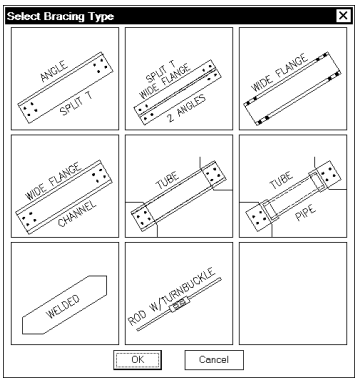
NOTE:

- For double angles there will be a minimum of two fillers for braces over 6'-0" long and a minimum of one filler for braces less than 6'-0". The 6'-0" control is from hole to hole.
- Dimensions A1, A2, A3, and A4 may be 0 (zero).
- Number of rows of bolts may be 0 (zero).
- Enter 5 as the same value as 5a and negative to calculate a gusset with a single line of holes offset from the work line.
- The number of bolts may be different at each end of the brace. This is to allow one end to be bolted and the other end to be welded.
- If the end information is read from files, one end may have a single line and the other end may have a double line of bolts.
- Offsets #8 and #9 for drawing braces may be entered as positive or negative. A positive dimension will offset the work points in the direction shown and a negative dimension will offset the work points in the opposite direction.
- #1 can be 0 to draw a single brace in the vertical position.
- #2 can be 0 to draw a single brace in the horizontal position.

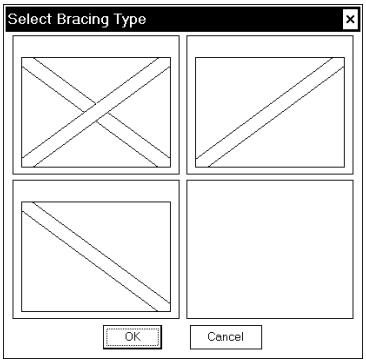
Welded Brace

This is for drawing a brace with the ends mitered to fit against other members in a rectangular frame.

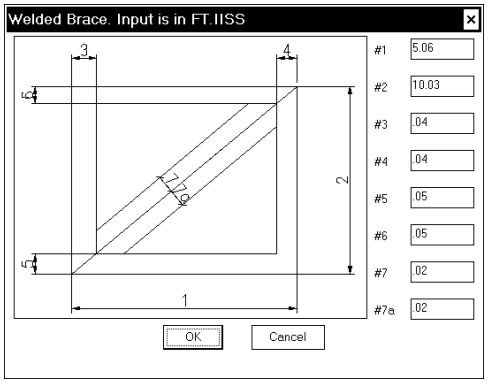
`Command:` Pick [AutoSD or AutoSD II] > Bracing > Draw > Brace



An icon menu appears.
Pick Welded



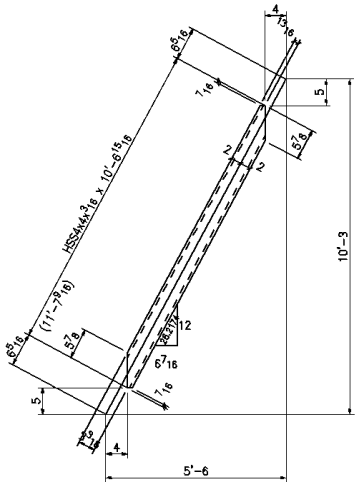
Another icon menu appears.
Pick the type of brace



A dialog box is used to enter dimensions.

If the lower end is on a horizontal support and not at a corner, enter #3 as a negative number that would cause the vertical line for #3 to fall to the left of the extreme end point of the brace. This will give a single cut at the end of the brace. This same technique can be used at each end for #3, #4, #5 and #6.

Member size is selected from a dialog box.

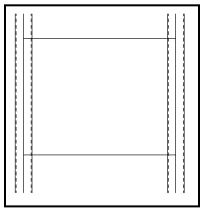
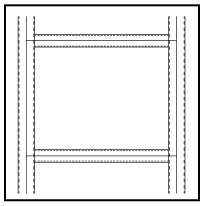
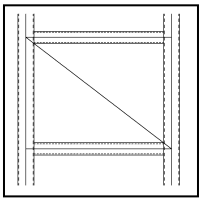
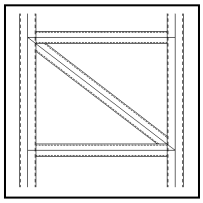


After entering the dimensions and selecting a member size and insertion point the brace is drawn and billed out.

1D1
ONE

HSS in Frame

Command: Pick [AutoSD or AutoSD II] > Bracing > Draw > HSS in Frame

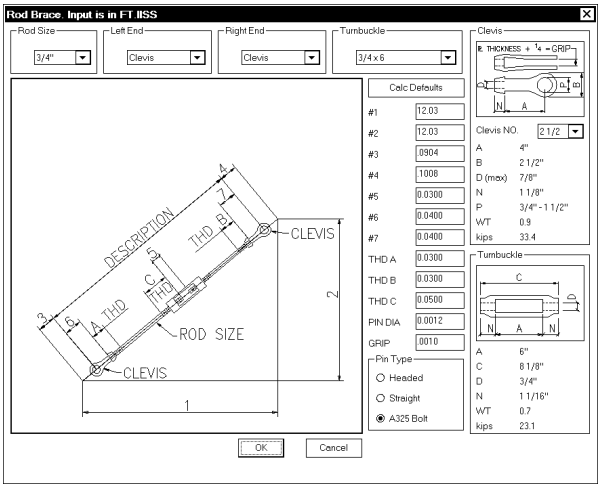





Step 1 Step 2 Step 3 Step 4

This command is for drawing the tube members in a welded frame. Start with the frame as shown in Step 1 with the continuous members and the centerlines of the cross members drawn. Pick the "HSS in Frame" command and pick a cross member centerline. Enter the width of the tube for the cross member. It will be drawn between the continuous members as shown in Step 2. Complete all of the cross members and then draw the centerlines of the diagonals as shown in Step 3. Pick the "HSS in Frame" command and pick a diagonal centerline. Enter the width of the tube for the diagonal member. It will be drawn between the continuous and cross members as shown in Step 4.

Rod Brace

Command: Pick [AutoSD or AutoSD II] > Bracing > Draw > Brace > Rod w/Turnbuckle

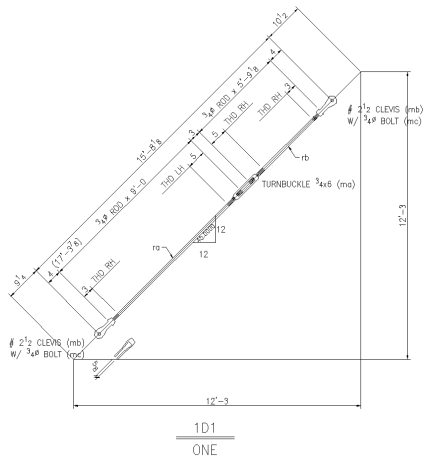


Rod Brace can detail a two piece rod brace with a turnbuckle. The ends can have a clevis, 1 nut, 2 nuts, a nut with a hillside washer or be left plain. Rod size is from 3/8" to 5". Select a rod size. The turnbuckle list is changed to match the rod size, data on the turnbuckle selected is given in the lower right corner of the dialog box and a suggested clevis size along with its data is give in the upper right corner. 3/8" - 3/4" rods only have one clevis size but 7/8" and larger have more than one size available.

Select the end conditions and a turnbuckle length. If a clevis is used and more than one is available select the one needed. Now click on the "Calc Defaults" button. This will fill in some of the entries based on the end condition and turnbuckle selected. Verify these values and enter the remaining entries. #1 can be 0 to draw the brace in the vertical position. #2 can be 0 to draw the brace in the horizontal position. Select the pin type and pick OK to draw the rod brace.

```
Insertion point at lower left end:
              Main mark: 1D1
Are marks RIGHT and LEFT (Yes/No) <N>: Enter
```

```
              Quantity: 1
Sub assembly mark for rod @ left end: ra
Sub assembly mark for rod @ right end: rb
Sub assembly mark for Turnbuckle: ma
Sub assembly mark for Clevis: mb
Sub assembly mark for Clevis Pin: mc
Shop bill information on mark 1D1
Place shop bill text on LINE <4>: Enter
Shop bill information on mark ra
Shop bill information on mark rb
Shop bill information on mark ma
Shop bill information on mark mb
Shop bill information on mark mc
```



Parallel Brace Tower

Command: Pick [AutoSD or AutoSD II] > Bracing > Parallel Brace Tower

Select 2 or 4 sides. If 4 sides are selected, H will be the vertical height and a new height will calculated for the true length. Enter values for B, H, T and select the number of panels. Pick Calc to get the results.

The results may be viewed one panel at a time. Pick Draw to draw the tower with all dimensions.

Picking OK will close the dialog box.

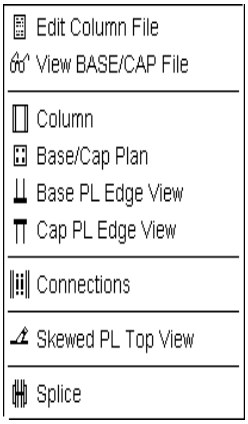
Calculate Gusset Plate (Truss)

Command: Pick [AutoSD or AutoSD II] > Bracing > Calculate Gusset PL (Truss)

Select the plate type and orientation. Select a diagonal and vertical member and whether there is 1 or 2 required. If the member is a pipe or tube the program defaults to only one member required. Enter the load on each member or enter 0, (zero), to use the area for calculating the load in tension and select a grade of steel. Enter dimension “A” to check an existing plate detail or leave it blank to let the program calculate the minimum for “A” with the clearance given for #4. Enter all dimensions required and pick Calculate to get the answers. Pick Print to print the answers on the drawing using a leader.
The loads on the output, and used to calculate the welds, is the greater of the load entered and the load calculated based on the area of the members in tension.
Output “A” is the greater of “A” from the input column and the dimension calculated to achieve the given clearance.
WS = length of weld on diagonal member.
WV = length of weld on vertical member.
WH = length of weld on horizontal edge of plate.

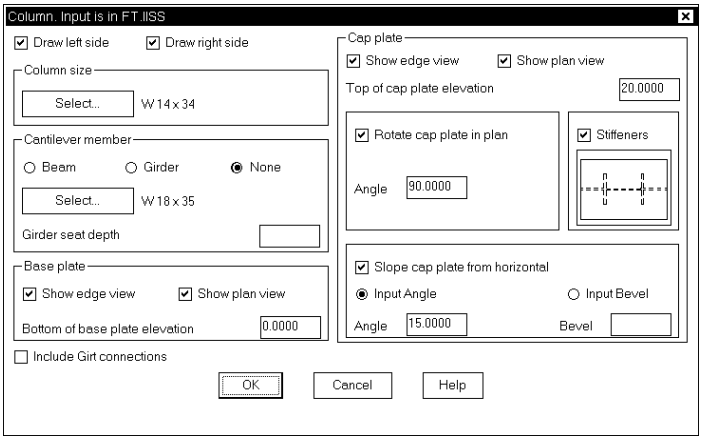
Chapter 9 - Columns

All programs in this chapter are under the AutoSD > Column pulldown menu, Columns toolbar and AutoSD II > Columns ribbon menu.

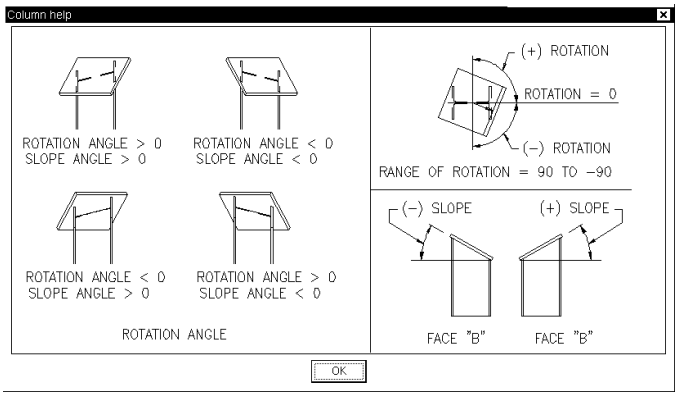


This program is for detailing columns in a vertical position only. Columns can be detailed from a column file or they can be detailed by supplying the connections manually. A column file can be created when the beams are detailed that frame into a column or it can be created from the Elevation Generator or it can be created from scratch using the "Edit Col File" program. All column file names have an extension of .col and will be saved in your job folder. Column files must be edited before they can be used if they were created when detailing beams or from the Elevation Generator. The least you have to do is Pick OK when the "Column Connection" program opens. You should, however, check the file to be sure all connections are listed and add holes for any bracing connections. If you edit a file and later add another connection from the beam program the "Column Connection" program will launch when you detail that column in order to update the column file. If you have wing plates in the web with stiffeners, the stiffeners must be listed first if you want the wing plate to extend to the stiffeners.

Splices and any additional connections can be added after the column is drawn. Chain dimensions and tail dimensions are calculated and placed on the drawing for you.



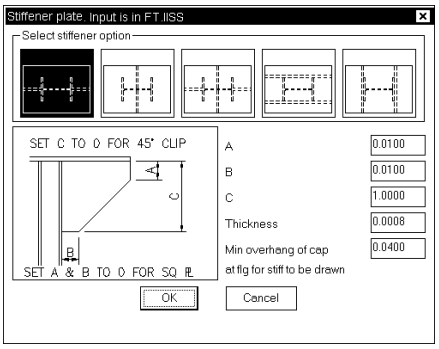
Draw left side: Checking this will draw face "A".
Draw right side: Checking this will draw face "C".
Column size: Pick the button to change the size of the column. You can detail wide flange, channel, pipe and tube columns.
Cantilever member: If you have a beam or girder spanning across the top of the column you can enter the depth of the beam or girder. Picking the "Select" button allows you to select a beam size and will fill in the beam depth for you.
Base plate: Base plates can be put on by the column program or they can be added later. Checking "Show edge view" will draw a base plate at the bottom of the column and give you the option to draw the plan view. Leaving it unchecked will cause the column to be drawn without a base plate. Base plates can be rotated when they are drawn.
Cap plate: Cap plates can be put on by the column program or they can be added later. Checking "Show edge view" will draw a cap plate at the top of the column and give you the option to draw the plan view and add stiffeners. Leaving it unchecked will cause the column to be drawn without a cap plate or stiffeners.



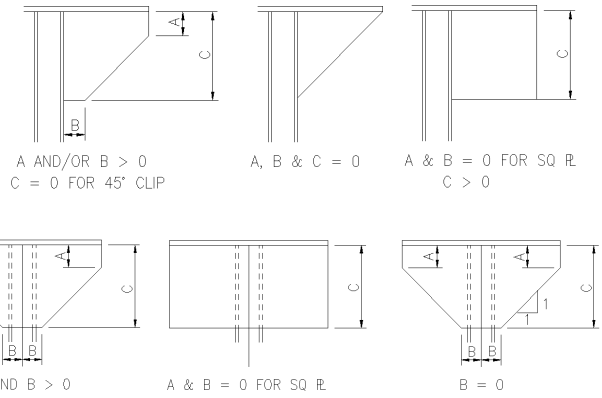
Rotate cap plate in plan: Cap plates can be rotated to any angle between 90 and -90.
Slope cap plate from horizontal: Cap plates can be sloped to any angle between 75 and -75. or to any bevel between 45 and -45. Picking the Help button will open the dialog box shown to the left.

Include Girt connections: Checking this option will include a string of dimensions on the right side for girts and add girt connections to the column connection dialog box.

Stiffeners:



Pick the image under Stiffeners to open the options for stiffeners. The pictures to the right show the various shapes you can have for stiffener plates. A stiffener will not be drawn on the flange of a "W" shape or any side of a tube or pipe if the cap extends less than the value entered for the minimum overhang.



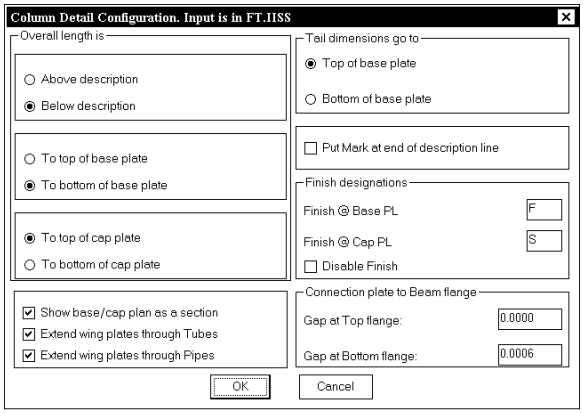
Column Grid | Reference: After entering the main mark and quantity you are prompted for the Column Grid | Reference. This is optional information that will be placed under the main mark. Pressing the enter key without supplying any text will omit this option. Entering text with the "|" character, as in "150|11", will place the "150" grid number over the "11" reference number separated by a line.

If you draw the column by supplying the connections manually, you have the option of drawing faces A and C. If you do not draw face A or C and have a connection in face A and B at the same elevation, the dimensions will overwrite each other. You should draw a face if you have any connections in that face. Face A is the left flange, B is near side web, C is right flange and D is far side web. Start with the lowest elevation and work your way up. When you see the prompt "Hole Spacing" or "From Elev. Down" or "From Elev. Up", enter the spacing in the format the drawing was set up to use, i.e. (FT.IISS or millimeters for metric). 3 rows of holes at a spacing of 4 1/2" with the first hole starting 3" below the elevation would be entered as .03,.0408,.0408. If you do not have any holes above or below the elevation just press enter at the prompt. Entering 0 will put a hole at the referenced elevation.

If you draw the column by using a column file, the program will draw face B and one or both of the other two faces. When prompted for the insertion point, a highlighted box will appear. The outline of the box will be the limits of the picture for the column. Pressing the "+" key will increase the height of the box making the column taller. At maximum height the column will be drawn to scale and the spacing of connections will be to scale. Pressing the "-" key will decrease the height of the box making the column shorter but it will not be made shorter than the originally calculated height. Increasing the height will give you more space between elevations. This box will enable you to position the picture on your sheet. If part of the box goes off the screen you should cancel the program. Zoom out and try again until the entire box is visible. When you have holes you will be asked for the "GAGE" and "MAIN GAGE". The "GAGE" is for the individual connection. The "MAIN GAGE" runs the length of the column and will be dimensioned above the top. If the "GAGE" and "MAIN GAGE" are different the connection will be dimensioned with the "GAGE".

Before detailing columns you should configure your customer. See page 14 for CUSTOMER CONFIGURATION.

Configuration options include location of piece mark, overall length to top or bottom of base plate, overall length to top or bottom of cap plate, and overall length placed above or below description. Extend wing plates through tube or pipe column, (Only for non-skewed plates).



BASE PL EDGE VIEW

CAP PL EDGE VIEW

These two programs will draw the edge view of a plate on an existing column. They are affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.

BASE/CAP PLAN

This is for creating base/cap plates and drawing the plan view.

Symmetrical options: A check in these boxes will automatically fill out the "symmetrical" value when the first value is entered.

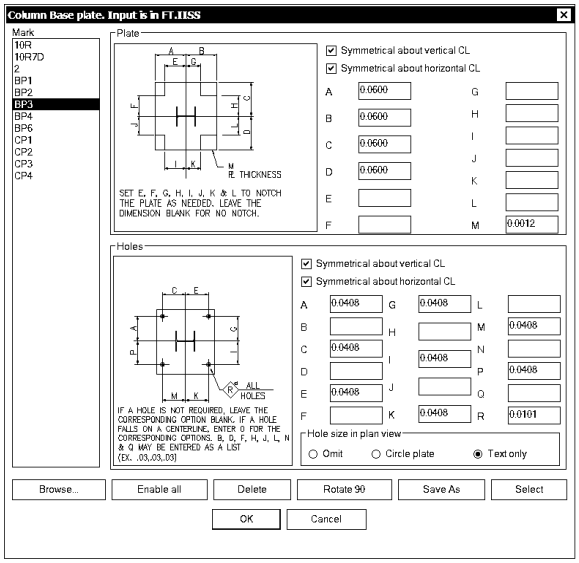
Browse: Look in other folders for the basecap.pl file.

Enable all: Turns on all input boxes.

Delete: Deletes the select mark.

Rotate 90: Rotates all input so plate will be rotated counter-clockwise 90 degrees.

Save As: Saves current plate as a different mark.



Select: Opens a dialog box for selecting standard plate configurations. Input boxes will be disabled to make it easier to input for a particular plate type and hole pattern.
Hole size in plan view: These options are for noting the hole size. "Omit" will not put a hole size. "Circle plate" will put a rectangular hidden line around the plate with a hole size. "Text only" will note the hole size under the plate size. This setting is separate for base and cap plates.

The scale is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command when drawing a plate.

VIEW BASE/CAP FILE

Base and cap plates are both saved in a file called BASECAP.PL and is located in your customer job folder. Descriptions can be replaced, added to, edited and deleted from this file. This dialog box is also used by other programs to select a saved plate for detailing.

If you edit a description, picking ***Cancel*** will cause you to lose all changes. Picking ***OK*** will save all changes to the file. Picking ***Browse*** will allow you to select a basecap.pl file or an older cap_base.pl file from another folder. Picking ***Enable All*** will enable all options and set the hole pattern to the default. ***Delete*** will remove the current selection. ***Rotate 90*** will relocate all dimensions to rotate the plate counter clockwise 90^{o}. ***Select*** lets you select a preset plate configuration. ***Save As*** will save the current selection as another mark. Plate marks can be saved as upper or lower case but is not case sensitive. You cannot save a mark pa and a mark PA. The mark you use to save the plate as does not have to be the piece mark used in the detail but it is the default.

There are two sets of input, one for the plate and one for the holes. Plate dimensions A, B, C, D and M must be entered. All other dimensions are optional. Leave the dimension blank if that option is not needed. Dimensions B, D, F, H, J, L, N and Q for Holes may be entered as a single dimension or as a list. Lists are dimensions separated by commas. Example: .03,.03,.03. Leave all dimensions blank if you do not want a hole at that location.

If the plate is symmetrical about the horizontal or vertical center line you can put a check in the box for that option. Enter the dimension for one side and the other side is entered for you when you leave that box by pressing the tab key. If you check both horizontal & vertical, enter the dimensions in one corner and then "tab" through the others to fill them in automatically.

The rotation of the bolt pattern is based on the column web being horizontal.

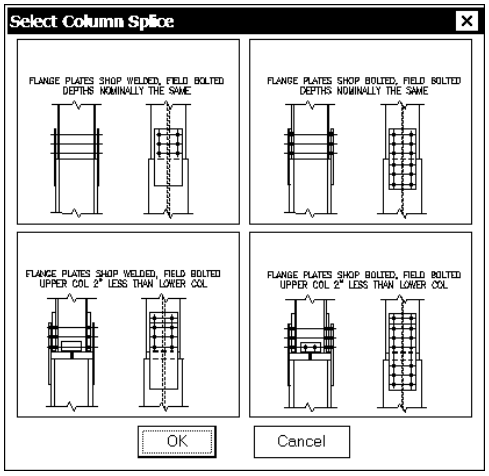
SKEWED PL TOP VIEW

This is similar to the program of the same name under the AutoSD > Beams pulldown menu. The program for columns can draw the end view of the column with the plate drawn at any angle, including 0 and 90. The column can be any wide flange, pipe or tube. The scale is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.

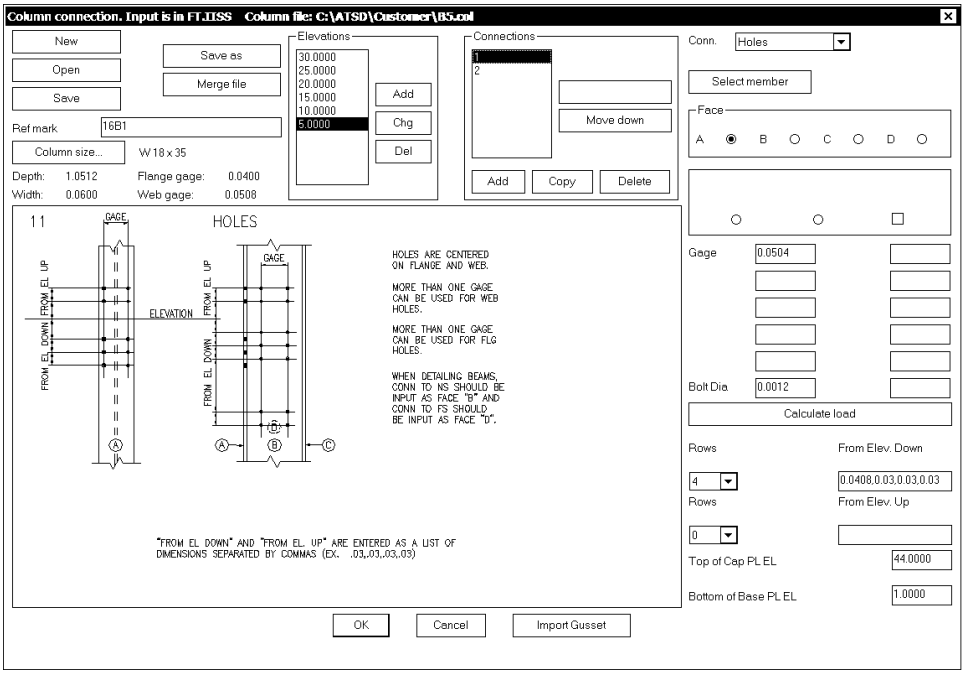
SPLICES

There are two types of splices. One is for columns that are nominally the same depth and the other is for columns that are nominally 2" different in depth. Of these two types you can have the flange plates shop bolted or shop welded. An icon menu is used to choose the type of splice required. Splices are for "C/HP/M/MC/S/W" shapes only.

You can splice any size column from 4" to 44".
This is the icon menu for splices.



EDIT COLUMN FILE



This program is used to process or edit an existing column description file or create one from scratch. When you select "Edit Col File" from the AutoSD > Columns pulldown menu this program is launched.

Use your mouse to select items in the program or use the tab key to move forward and shift + tab to move backward. If you have created a description file from the beam program pick OPEN. Change the directories box to your customer job folder, select the file and then select Open or double click on the file.

A list of elevations is shown in the Elevations box at the top of the screen. They will be listed in order with the lowest elevation at the bottom. The number of connections at a selected elevation is shown to the right of the Elevations in the Connections box. You change from one connection to another at the same elevation by selecting a different number in the Connections box. The FACE box indicates which face the connection is on. Face A is the left flange, B is near side web, C is the right flange and D is far side web.

New

Creates a new empty column file.

Open

Opens an existing column file.

Save

Saves the current column file.

SAVE AS

This will save the current file under a different name giving you a copy. This is useful when you have two columns that are similar. After completing the description file for one column, SAVE it for that column and then SAVE AS a different file name for the other column. The program will switch to the new copy and you can make the changes needed.

MERGE FILE

This is used to merge two column files together. This is useful when you have more than one person working on beams that frame into common columns or when you create column files from the Elevation Generator.

Column size

Select the column size being detailed. The depth, width, flange gage and web gage are displayed for "W" & "C" shapes.

Elevations box

ADD will add a new elevation with one connection called "NONE".
CHG will change the elevation of all connections at the selected elevation if the elevation you change to does not already exist. CHG will change only the elevation of the connection selected if you change to an existing elevation.
DEL will delete all connections and the elevation of the selected elevation.

Connections box

MOVE UP & **MOVE DOWN** will change the order if there is more than one connection at the selected elevation. Use to arrange wing plates and stiffener plates.
ADD will add a new connection to the selected elevation.
COPY will copy a connection from one elevation to another or to the same elevation.
DELETE will delete only the connection selected if there is more than one at the selected elevation.

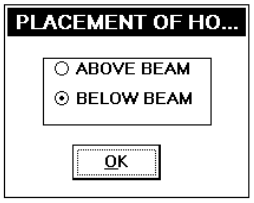
HINT: If you have connections differing in elevation by 2 1/2", change the lower elevation to be the same as the higher one and increase the first dim from elev. down by 2 1/2". This will reduce the clutter of dimensions when the column is detailed.

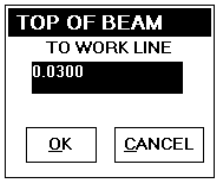
The large box in the middle of the screen shows the type and dimensions for the connection. Conn. is a pop down list of all available connections. Pick the box or the arrow at the right end and the list will pop down. Select the connection desired and the appropriate fields will be displayed below and a reference picture will be shown to the right. Below Conn. are the dimensions for the selected connection. Enter the dimensions in the format indicated in the title of the main screen. The example shown is FT.IISS. Press the tab key after each dimension and the cursor will jump down to the next dimension to enter. Dimensions for "FROM ELEV. DOWN" and "FROM ELEV. UP" are entered as a list of dimension separated by commas. Do not include any spaces. Example: .03,.03,.03,.03. Selecting a value under Rows will automatically fill in the list of dimensions for the number of rows selected.

If wing plates extend through a tube or pipe column you should list the plates that extend all the way through first. If you only have two connections at a given elevation and they are on adjacent sides, (90 degrees to each other), the connection listed first will extend through so it should have the most rows. If one of the plates is skewed it must be listed last. Use the Move up and Move down buttons to change the order.

If you want the wing plate in the web to extend upward and downward to weld to stiffener plates, the stiffeners should be listed first and the wing plate second. If the wing plates are listed first they will not extend to the stiffener plates.

IMPORT GUSSET



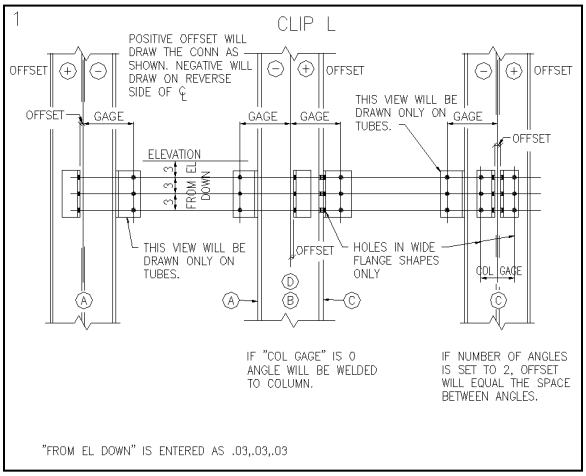


This is used to add the hole spacing for a gusset plate connection that uses clips on the gusset to connect to a column. When selected, a file and folder dialog box pops up. After selecting the file another box pops up asking for the placement of holes. Select above or below beam and then select OK. If you select below beam another box pops up asking for the location of the work line. Enter the dimension and select OK.

There are 16 different connections available to the column program. 15 are shown below. The 16th is “NONE”. It puts an elevation line with no connections for use in allowing space for adding a detail later. Start by picking AutoSD > Columns > Connections. After completing one connection you can start another one by pressing Enter.

CONNECTIONS

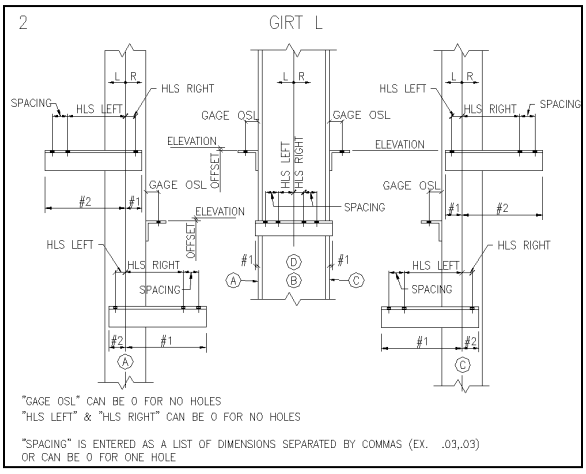
(1) Clip Angle



GAGE is from center of column to center of holes.
OFFSET is from center of column to face of clip. It is the gap between clips if you have 2 clips.
COL GAGE is the gage of the column. Enter 0 if the clips are shop welded to the column.

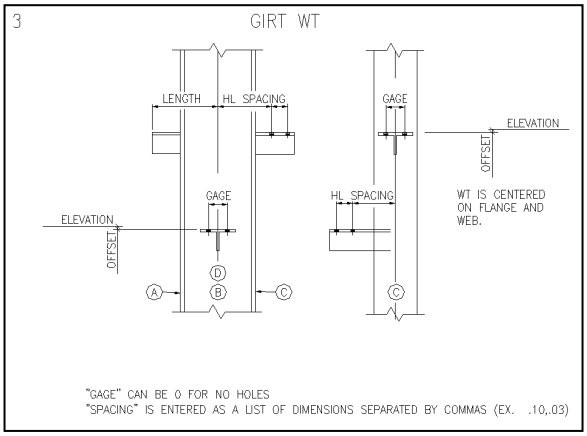
FROM ELEV DOWN is the location of holes below the given elevation. Use the Rows pop down list to select the number of rows needed.
FROM ELEV UP is the location of holes above the given elevation. Use the Rows pop down list to select the number of rows needed.

(2) Girt Angle

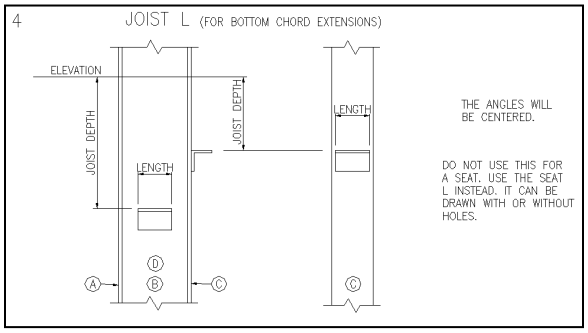


GAGE OSL is the gage of the outstanding leg. This can be 0 if no holes are required. OFFSET is from the elevation line down to the heel of the angle. HLS RIGHT is from center of column to center of first hole to the right. HLS LEFT is from center of column to center of first hole to the left. #1 and #2 is how far the angle will extend from the center line. HOLE SPACING is the spacing between holes. ANGLE IS TOED Up or Down is the direction of the vertical leg.

(3) Girt WT

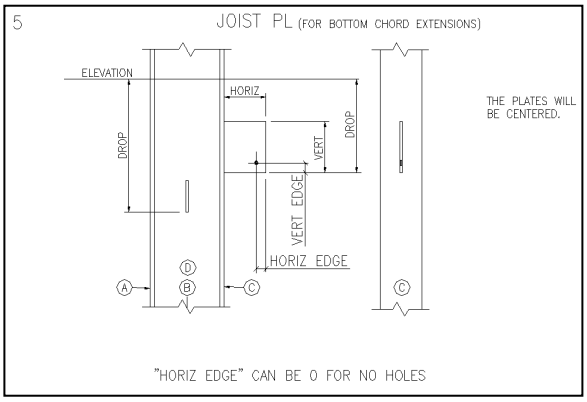


GAGE is the gage of the WT flange. This can be 0 if no holes are required.
OFFSET is from the elevation line down to the flange of the WT.
LENGTH is the length of the WT from the column center line.
HOLE SPACING is the spacing of holes from the center of the column.
STEM IS TOED Up or Down is the direction of the stem.



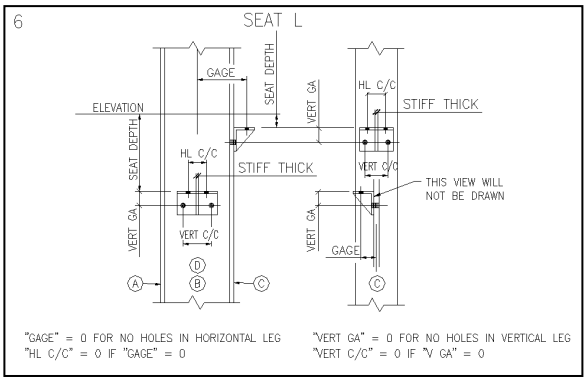
(4) Joist Angle

This is a joist bottom chord stabilizer angle. Do not use this connection for a seat.
JOIST DEPTH is the distance from the elevation to the top of the angle.
LENGTH is the cut length of the angle.
ANGLE IS TOED Up or Down is the direction of the vertical leg.



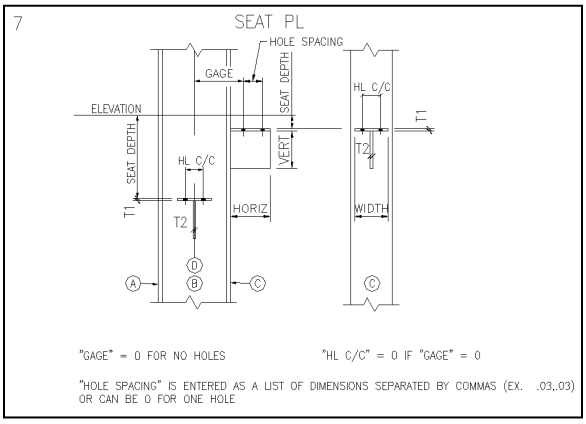
(5) Joist Plate

This is a joist bottom chord stabilizer plate.
HORIZ is the horizontal size of the plate.
VERT is the vertical size of the plate.
THICKNESS is the thickness of the plate.
HORIZ EDGE is the horizontal edge distance for the hole.
VERT EDGE is the vertical edge distance for the hole.
DROP is the distance from the elevation down to the bottom of the plate.



(6) Seat Angle

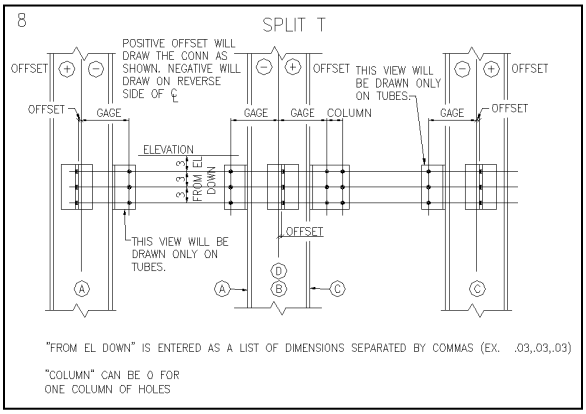
GAGE is the distance from the center of the column to the hole. This can be 0 if no holes are required.
HL C/C is the center to center of holes.
VERT GA is the gage of the vertical leg. This should be 0 if the angle is welded.
VERT C/C is the C/C of holes in the vertical leg.
LENGTH is the cut length of the angle.
SEAT DEPTH is the distance from the elevation down to the top of the angle.
STIFF THICK is the stiffener thickness if the stiffener option is selected.



(7) Seat Plate

GAGE is the distance from the center of the column to the hole. This can be 0 if no holes are required.
HL C/C is the center to center of holes. This should be 0 if GAGE is 0.
HORIZ DIM is the horizontal size of the stiffener.
VERT DIM is the vertical size of the stiffener.
WIDTH is the width of the horizontal plate

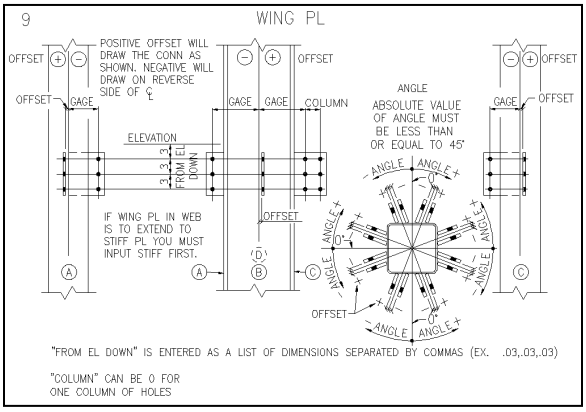
SEAT DEPTH is the distance from the given elevation down to the top of the horizontal plate.



(8) Split T

Split t can be WT, MT or ST.
GAGE is from the center of the column to the first hole.
OFFSET is from the center of the column to the side of the stem.
COLUMN is the distance to the 2nd column of holes from the 1st. This is 0 or blank for one column.

FROM ELEV DOWN is the location of holes below the given elevation.
FROM ELEV UP is the location of holes above the given elevation.



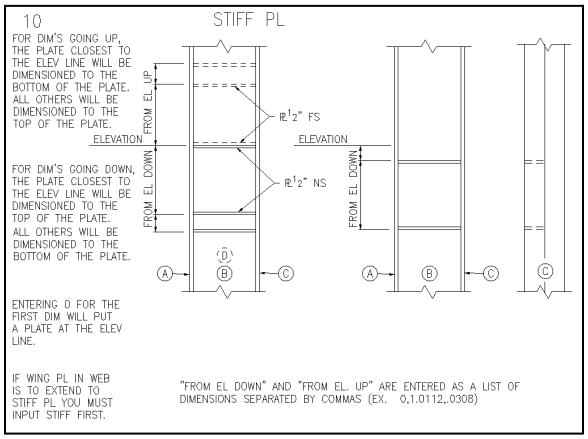
(9) Wing Plate

GAGE is from the center of the column to the first hole.
OFFSET is from the center of the column to the side of the plate. Positive goes to the right, negative to the left as viewed facing the side of the column the plate is on.
THICKNESS is the thickness of the plate.
COLUMN is the distance to the 2nd column of holes from the 1st. This is 0 or blank for one column.
ANGLE is the angle from horizontal if the plate is on face A or C. It is the angle from vertical if the plate is on face B or D.

FROM ELEV DOWN is the location of holes below the given elevation.
FROM ELEV UP is the location of holes above the given elevation.

If wing plates extend through a tube or pipe column you should detail the second plate that has a connection on the opposite side immediately after the first wing plate. If you only have two wing plate connections at a given elevation and they are on adjacent sides, (90 degrees to each other), the connection detailed first will extend through so it should have the most rows. If a wing plate is skewed it must be detailed last.

If you want the wing plate in the web to extend upward and downward to weld to stiffener plates, the stiffeners should be entered first and the wing plate second. If the wing plates are entered first they will not extend to the stiffener plates.

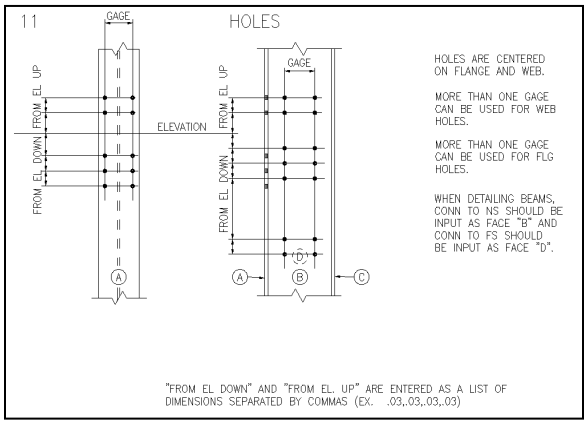


(10) Stiff Plate

THICKNESS is the thickness of the plate.

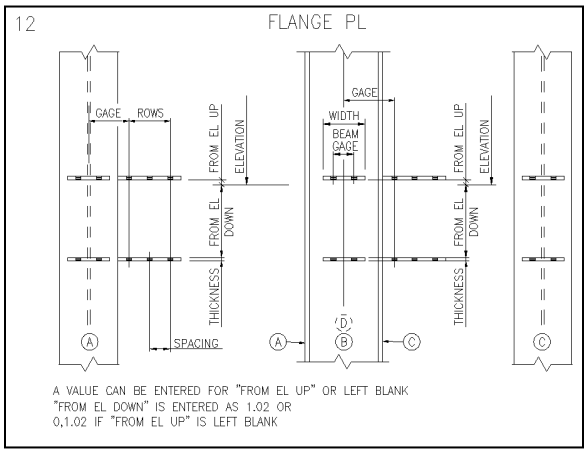
FROM ELEV DOWN is the location of stiffeners below the given elevation. 0 will put a stiffener at the elevation.

FROM ELEV UP is the location of stiffeners above the given elevation. 0 will put a stiffener at the elevation.



(11) Holes

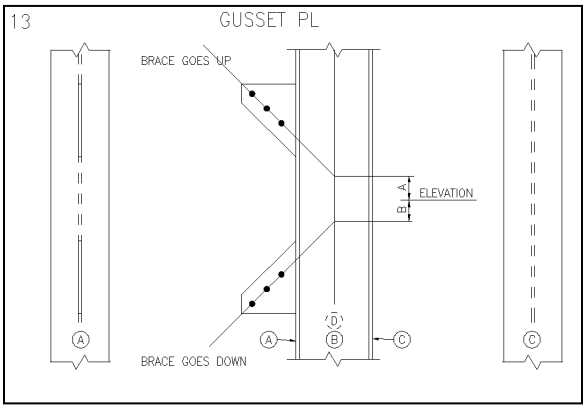
GAGE is the horizontal center to center of holes.
FROM ELEV DOWN is the location of holes below the given elevation. 0 will put a hole at the elevation.
FROM ELEV UP is the location of holes above the given elevation. 0 will put a hole at the elevation.



(12) Flange Plate

GAGE is the horizontal distance from center of column to center of the first hole.
BEAM GAGE is gage of the beam that would bolt to the plate.
THICKNESS is for both plates in this connection.
ROWS is the number of rows of bolts in both plates.
SPACING is the space between each bolt in both plates.
WIDTH is the width of the plate.

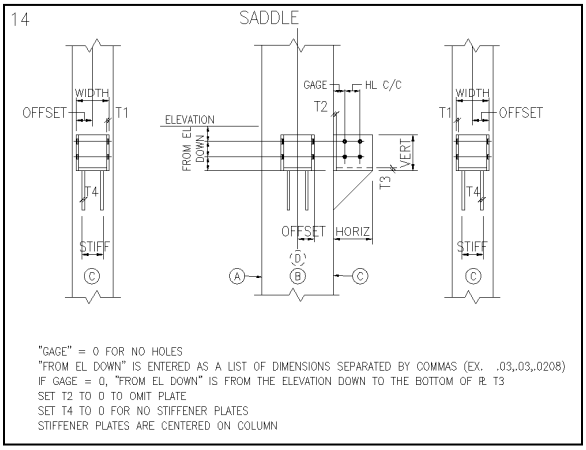
FROM ELEV DOWN is the location of plates below the given elevation. 0 will put the bottom of a plate at the elevation.
FROM ELEV UP is the location of plates above the given elevation. 0 will put the bottom of a plate at the elevation.



(13) Gusset Plate

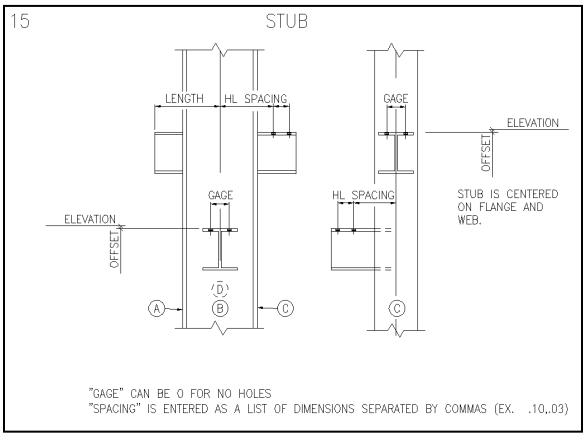
(Must have Bracing package)
Work line of brace goes up or down.
Select the one that suits your brace condition.

"**A**" is the distance from the elevation up to the work point. "**B**" is the distance from the elevation down to the work point. You cannot have both for one connection. 0 will put the WP on the elevation.



(14) Saddle

GAGE is the distance from the face of the column to the holes. This can be 0 if no holes are required.
HL C/C is the center to center of holes.
HORIZ is the horizontal length of the plates.
VERT is the vertical length of the plates.
WIDTH is the overall width of the saddle.
OFFSET is the distance from column centerline to one side of the saddle.
STIFF DIM is the out to out distance if the two stiffener option is selected.
PL THK T1, T2, T3 & T4 are the thickness of the plates.
FROM ELEV DOWN is the location of holes below the given elevation. If GAGE is 0, FROM ELEV DOWN is the location of the bottom of the saddle from the elevation.



(15) Stub

GAGE is the gage of the flange. This can be 0 if no holes are required.
OFFSET is from the elevation line down to the top of the stub beam
LENGTH is the length of the from the column center line.
HOLE SPACING is the spacing of holes from the center of the column.

Chapter 10 – Shop Bill

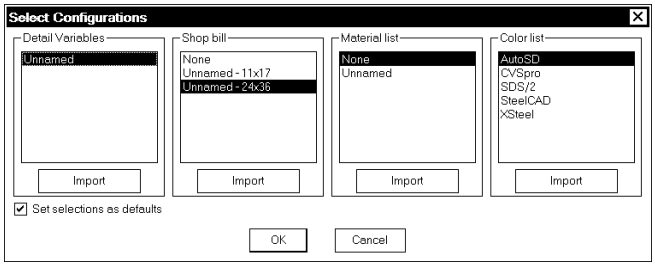
BOM

Pulldown: AutoSD > Shop Bill > Shop Bill Text / Attributes
Ribbon: AutoSD I > Shop bill

The shop bill program will make it easy to fill out the shop bill using text or attributes. Text is easier to work with but attributes will allow you to extract the data to an ASCII file where it can be used by other programs. The shop bill program can be easily configured for each customer that you detail for and you can have as many configurations as you need.

Start a new drawing using the AutoSD.dwt template and run "setup", selecting Architect, 1" = 1' and D- 24x36. Now select AutoSD > Shop bill from the pulldown menu and then “Insert Shop Bill”. Change folders to \AUTOSD\CUSTOMER and select 24x36BOM.dwg. A typical shop bill appears. (For METRIC, select a scale of 1:12 and paper size of D- 24x36. select the 24x36BOM.dwg file located in \AUTOSD\BONUS\METRIC folder.) You can copy these files to your own job folder and edit them or create your own from scratch.

Command: Pick [AutoSD or AutoSD I] > Shop bill > Shop Bill Text / Attributes
A dialog box pops up.
Select “Unnamed - 24x36” under Shop bill and pick OK.




```
Place shop bill text on Line <1>: 5
Main mark: 1B3
Are marks RIGHT and LEFT (Y/N) <N>: Enter
Quantity: 6
Quantity for sequence 1: 2
Quantity for sequence 2: 2
Quantity for sequence 3: 2
Description: W14x22
(FT.IISS) Length: 13.0508
Remarks:
Command: Enter
Line <6>: Enter
Main mark: Enter
Sub assembly mark: pa
Quantity: 6
Description: `P`3x3`4
(FT.IISS) Length: 1.0208
Remarks: Enter
Command: Enter
Line <7>: Enter
Main mark: Enter
Sub assembly mark: aa
Quantity: 12
Description: L3`8x3`2x`5
```

Size of beam.
13'-5 1/2
Text is put in shop bill.
To start program again
To accept line #6.
Toggles past main mark.
Sub mark

PL 3/16x3 1/4
1'-2 1/2

To start program again
To accept line #7.
Toggles past main mark.
Sub mark

L3 1/2x3 1/2x5/16

```
(FT.IISS) Length: .0808                    0'-8 1/2
Remarks: Enter                             Finished
```

Zoom in to the top of the shop bill and see what happened. The text is there.

SEQ 6	SEQ 5	SEQ 4	SEQ 3	SEQ 2	SEQ 1	UNIT QTY	QUANTITY	SHIPPING MARK		DESCRIPTION	FEET	INCHES	ASSEMBLY MARK	UNIT WEIGHT	TOTAL WEIGHT	
																1
							3	1B1		W14X22	22	3 1/2			1471.25	2
																3
																4
			2	2	2		6	1B3		W14X22	13	5 1/2			1776.50	5
									6	PL 3/16X3 1/4	1	2 1/2	pa		15.02	6
									12	L3 1/2X3 1/2X5/16	0	8 1/2	aa		61.20	7
																8
																9
			1	9	2		12	1B5		W10X12	8	5 1/4			1215.00	10
																11

The shop bill is placed on a layer called shopbill. The portion of the shop bill for sequences is on a layer called sequence. This layer can be frozen if sequences are not being used.

CREATING THE SHOP BILL

Quit the drawing you are in and begin a new drawing. The name of the drawing will be 24x36BOM and is saved in the folder for this customer. "24x36BOM" is for a 24 x 36 size shop bill drawing. A different drawing will need to be created for each paper size but one paper size can be used for any scale as long as the original drawing is created at a scale of 1” = 1’-0. You can use any legal drawing name allowing you to have more than one shop bill drawing in a single folder. You can also create an erection sheet and insert it the same as you would a shop bill sheet. Once inside the drawing editor, select SETUP then Archtect, 1"=1' and D- 24x36. Use 1:12 scale for metric units.

Create a new layer called SHOPBILL and set it as current.

```
Command: LAYER
Pick New
Change layer name to SHOPBILL
Pick Current
Pick OK
```

Now it is up to you to draw the shop bill using AutoCAD’s drawing commands. The only restriction is the drawing must be setup to a scale of 1"=1' or 1:12 for metric.
Put the headings at the top of the shop bill and numbers to the left of each line for easy reference. If you do not want the line numbers to plot put them outside the right border line.

Create a new layer called SEQUENCE and set it as current.

```
Command: LAYER
Pick New
```

```
Change layer name to SEQUENCE
Pick Current
Pick OK
```

Now draw the sequence portion of the shop bill. You can have as many as six columns. See page 152 for an example of the top portion of a typical shop bill.

The sequence portion is placed on a separate layer so that it can be frozen or thawed as needed without having to create a different shop bill drawing. This layer will be thawed automatically if the shop bill configuration includes at least one sequence. It will be frozen automatically if the shop bill configuration does not include any sequences.

Now that you have the shop bill drawn, let's change the color of the lines to red. You may use any color you want.

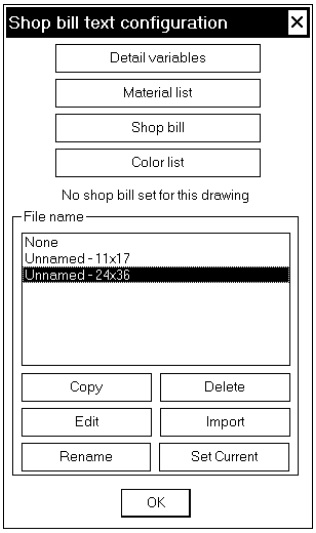
```
Command: CHANGE
Select objects:                          Pick all lines but not the text
# selected, # found                      Do not pick the border.
Select objects:
PROP
Properties /<change point>: COLOR
New entity color <BYLAYER>: RED
Properties /<change point>: Enter
```

CONFIGURING SHOP BILL

The items that can be configured are as follows:

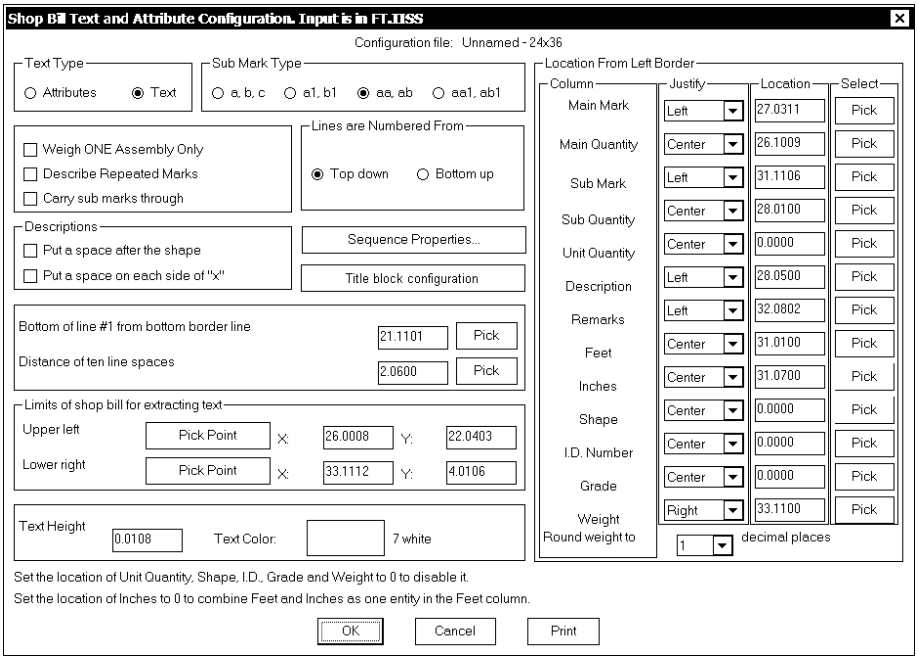
- Quantity of main mark
- Main mark (1B1, 3C1, etc.)
- Quantity of sub assembly mark
- Sub assembly mark (aa, pa, ma, etc.)
- Unit quantity for sub marks
- Description (main piece and sub piece)
- Length in feet or mm
- Length in inches (For architectural and decimal units only)
- Shape designation (W, L, PL, etc.)
- Mill order I.D. number for main mark
- Grade
- Remarks
- Weight
- Sequence quantity for main mark

Pick AutoSD > Customer Configuration from the pulldown menu and pick Shop bill to change the settings in a configuration. Select the file you want to change and pick Edit. Picking “Set Current” will save the name of the selected file in the drawing and that file will be used until another one is set.



The first time you open the configuration box you will have one file in it called “None”, one called “Unnamed - 11x17” and one called “Unnamed - 24x36”. You can edit and use the last two. You can rename them or copy them and make more files. You can delete all files except for 2. “None” is one that cannot be deleted. If you have an older version that used the sbtext.cfg file you can import those files and use them. Older versions had separate sbtext.cfg files saved in each job folder. This version has all of the configurations stored in one file called custbom.cfg located in your autosd folder. The metric version is called custbomm.cfg. You can also import individual configurations from other custBOM.cfg files.

After you pick Edit, another dialog box will pop up. You will need to select the options you want and edit the location of each column. You can use the "PICK" button to select the location of each column or you can enter the location in each edit box. All values are entered at a scale of 1” = 1’-0” for ARCH or 1:12 for METRIC regardless of the scale of the drawing. You will need to have the shop bill drawing open to use the pick feature accurately. If you want to make small adjustments in a column from a known location you may find it easier to adjust the value in the edit box.



Text Type:
Select if you want to use attributes or text.

Sub Mark Type:
Select the 2nd or last item here to use the sheet number as a suffix to the sub piece marks.

Weigh ONE Assembly Only:
Select this if you want the weight to be for one unit only.

Describe Repeated Marks:
Select this if you want the description filled out for all pieces with the same mark. Not selecting this will not put the description with a mark that has already been used in the shop bill.

Carry sub marks through: Selecting this will save new sub marks and description in the file submark.tbl in your customer job folder. When you use the same mark on following drawings the description from the file will be used.

Descriptions

Put a space after the shape:
Put a check in this box if you want to have a space between the shape designation and the rest of the description. When using "Shop Bill Text / Attributes" under "Beams" to enter a description in the shop bill, do not add a space after the shape if you have this box checked. A space will be added automatically. Putting one when you type the description will result in two spaces.

Put a space on each side of "x":
Put a check in this box if you want to have a space on each side of the "x" character in the description**.** When using "Shop Bill Text / Attributes" under "Beams" to enter a description in the shop bill, do not add a space before or after the "x" if you have this box checked. A space will be added automatically. Putting one when you type the description will result in two spaces.

Lines are numbered from
Select "Top down" or "Bottom up".

Limits of shop bill for extracting text:
Enter the X and Y coordinates of the upper left and lower right extents of text that will be in the shop bill. This is used to select the text using these two points as a window when you extract the shop bill information using text instead of attributes. Make sure the window does not include the headings or any line numbers.

Text Height and Text Color
Enter the height for text in the edit box. To change the text color, pick the color swatch and then select the color you want. The height and color selections affect attributes as well.

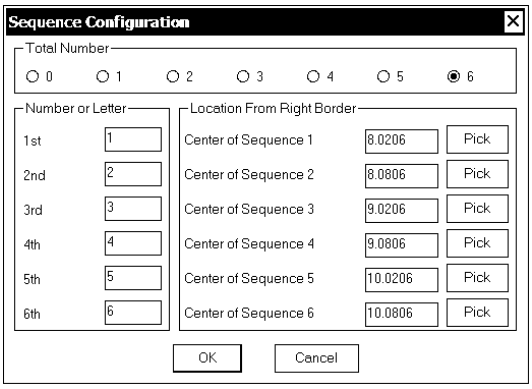
Print: Picking this will give you a print of all settings and options selected.

If the Weight category is non-zero you need to select whether the weight for sub pieces is for one assembly only or not. You have the option of extending the total weight of all assemblies or for just one.

Entering a 0 (zero) for UNIT QUANTITY, SHAPE, I.D., Number of SEQUENCES, GRADE or WEIGHT will cause those prompts to be deactivated and those categories will not be used. Entering a number other than zero will make them active. Entering a 0 (zero) for the location of inches will cause the feet and inches to be written as one number thus: 2'-8 1/2

Sequence Properties:

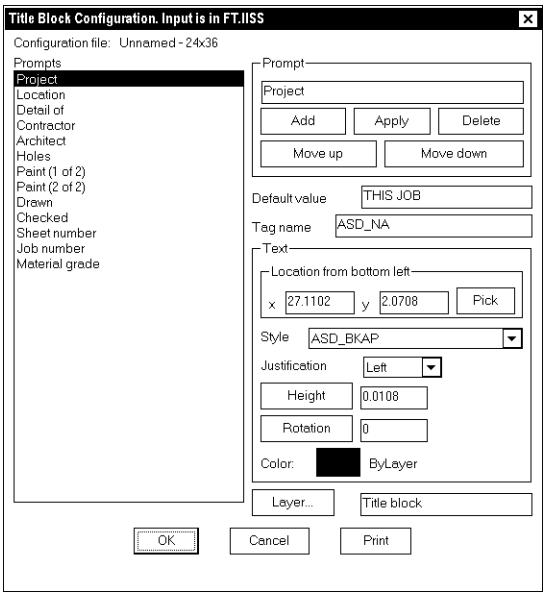
If you are going to use sequences you can have up to 6. Select the number of sequences under "Total Number". Under "Number or Letter" fill in the sequence designation you want to use in the prompt when asked for the quantity. Give the dimensions for the location for each sequence in place of the defaults shown or use the "PICK" feature to select the location of each column.



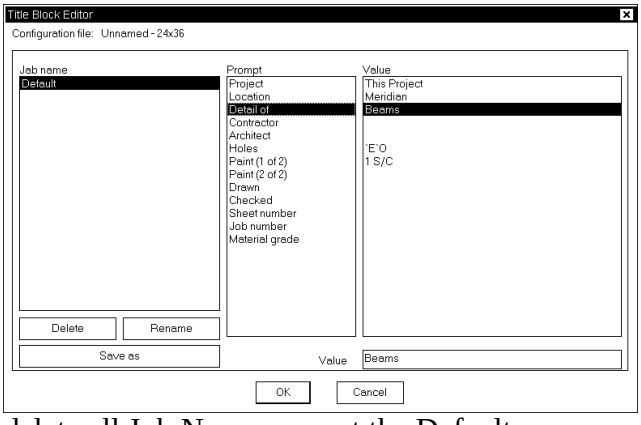
The configuration can be made at any scale but the shop bill must be drawn to a scale of 1"=1' for ARCH or 1:12 for METRIC.

Title block configuration

This will configure a block made of attributes for filling out the title block quickly and easily. The Title Block Configuration program allows you to add an attribute for each item that you want to include in the title block information. The “Move up” and “Move down” buttons will move the prompt in the prompt list but has no effect on where it is placed in the drawing. If you used the tag names ASD_NA for Project Name, ASD_NO for Job Number, ASD_GR for material grade and ASD_RE for the revision mark in your title block attributes, this information will be included in “.kss” file when extracting for Fabtrol.



Inserting the title block:



Pick AutoSD > Insert > Title block. You get the dialog box shown to the left. Pick one of the prompts and then pick the “Value” box at the bottom to change the value of the prompt. The value will be put in the title block and all changes will be saved when you pick the OK button. All values saved will become the defaults for the next time the title block program is opened. Pick “Save as” to make a copy of the selected Job Name. You can rename or delete all Job Names except the Default.

Once the title block is inserted, you can edit it with AutoCAD’s Attribute Editor by double clicking on one of the attributes, typing eattedit or pick Modify > Object > Attribute > Global.

ATTRIBUTES vs. TEXT

Even though AutoSD gives you the ability of extracting the bill of material using text, attributes allows you to extract the data to an ASCII file in a format that you control where it can be used by other programs not supplied by AutoSD. In order to extract information you must have a template file. The one shown below is supplied as an example and is used by the program extract.exe to convert the backward apostrophe (`) or tilde (~) characters to the corresponding fraction.

Attribute extract template file \AUTOSD\BONUS\ASD_SBL.TXT

ASD_MQ	C005000	(Main quantity)
ASD_MM	C011000	(Main mark)
ASD_SQ	C006000	(Sub assembly quantity)
ASD_SM	C007000	(Sub assembly mark)
ASD_DS	C035000	(Description)
ASD_FT	C003000	(Feet)
ASD_IN	C009000	(Inches)
ASD_SH	C004000	(Shape designation)
ASD_ID	C007000	(Mill order number)
ASD_SE	C004000	(No longer used but included to support older versions)
ASD_SP	C007000	(Material grade)
ASD_RM	C011000	(Remarks)
ASD_WT	C008000	(Weight)
ASD_S1	C010000	(S1 - S6 are sequence numbers for main marks.
ASD_S2	C010000	They are invisible and have no prompt.)
ASD_S3	C010000	
ASD_S4	C010000	
ASD_S5	C010000	
ASD_S6	C010000	
ASD_Q1	C004000	(Q1 - Q6 are quantities per sequence for main marks)
ASD_Q2	C004000	
ASD_Q3	C004000	
ASD_Q4	C004000	
ASD_Q5	C004000	
ASD_Q6	C004000	
BL:Y	N005000	(Y coordinate to enable proper sorting)

(The comments must not actually be part of the template file.) The file \AUTOSD\BONUS\SHOPBILL.TXT is used with version 4.0 and earlier and uses different attribute tag names. These older names are still recognized so you can still use your existing shop bill drawings. Each line has two fields. The first field is the tag name. The tag name is the name of one of the tags in the shop bill block. The second field contains the format for the data for that tag. All format fields start with a "C" denoting it as a character field except for the field BL:Y. It starts with a "N" denoting numeric. The next three digits are the field width characters. The last three are number of decimal places. All character fields have "000" for the

last three digits. Be sure to use a field width that is at least one character longer than actually needed. This will put a space between each field in the extracted file. If the field width is too small AutoCAD will truncate the output and the output may run together as one string, making processing of the output difficult.

CAUTION: The first fields in the attribute extract template file MUST be spelled as shown above. They may be in any order and some may be left out but the names MUST NOT be changed. **BL:Y N005000** must be left in the file and must be the last line.

NOTE: Do not use "tab" characters when constructing the template file. If you want to align the columns for readability, do so with ordinary spaces.

Each line of the extract file will contain information for each tag listed in the template file in the order they are listed. If all values are not used, those tags may be omitted from the template file. The order each line is written to the output file is the order that it was created in the drawing file. As long as the next line number that was used in the shop bill is higher than the previous, the output file will be in order. If you go back and add a line to a previous piece the output file will be out of order and that last line will be listed with the wrong piece in the extract file. This is the reason for the line **BL:Y N005000**. It outputs the Y coordinate of the block on each line. The lines can then be sorted using this number.

CAUTION: Template files and extract files use the extension ".txt". Do not give the same name and location for both or you will overwrite the template file with the extract file. Typically, the extract file name will be the same as your drawing. Use the following command to extract the attributes manually.

```
Command: ATTEXT
CDF, SDF or DXF Attribute extract (or Entities)? <C> S          Use the S option.
Template file: C:\AUTOSD\BONUS\ASD_SBL
Extract file name: C:\jobs\2012\1
```

The default output file name is the current file name, with a .TXT file extension.

EDIT SUB MARK LIST

Pulldown: AutoSD > Shop Bill > Edit Sub Mark List
Ribbon: AutoSD I > Shop bill

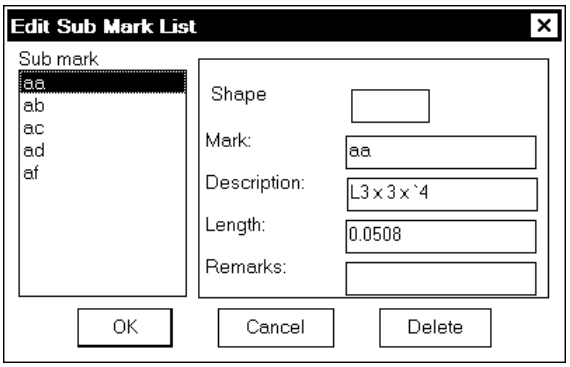
If you configure the shop bill to use the sheet number as a suffix to the sub piece marks, (selection 1 or 3 shown below), and check the box "Carry sub marks through" in the "Shop Bill Text and Attribute Configuration", the description for a mark will carry from one sheet to the next. Each time you enter a sub mark, the file submark.tbl in your customer job folder will be checked to see if the mark has been used before. If it has, the description, length and remarks will automatically be entered into the shop bill. If it has not been used you will be

asked to enter the description, length and remarks. After doing so they will be added to the list for future reference.

Line in shop bill configuration for selecting to carry sub marks through.
For sub marks use (0)a,b,c (1)a1, b1 (2)aa, ab (3)aa1, ab1 (0/1/2/3) <2>: **3**

Note: If you select option 0 and 2 the marks will be saved to and recalled from the submark.tbl file only if you end the mark with a number.
If the description, length or remark of a sub mark needs to be changed after it has been saved to the file submark.tbl you can change it with EDIT SUB MARK LIST located in the AutoSD > Shop bill pulldown menu.

A dialog box is used as shown to the right. Select a mark and the mark, description, length and remarks are displayed in edit boxes. To change one, click on an edit box and make the change. You can also delete a mark and the information associated with it. If you select Cancel, no changes will be saved.



EXTRACT SHOP BILL

BOM ATT
BOM TXT

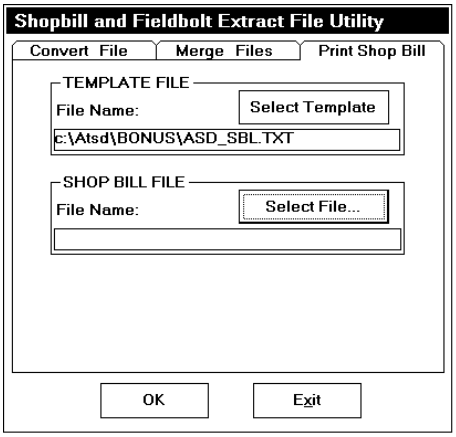
Pulldown: AutoSD > Extract > Shop Bill Using Attributes
Pulldown: AutoSD > Extract > Shop Bill Using Text
Ribbon: AutoSD I > Shop bill

Use this command from the AutoSD pulldown menu to extract the shop bill using text or attributes and run the program extractw.exe automatically to convert the extracted file using the template file ASD_SBL.TXT located in your AUTOSD\BONUS folder. (SHOPBILL.TXT is used with shop bills created with version 4.0 and earlier.)

Warning! Attributes must not be used if you are drawing in Paper space. You can use text and the text in the shop bill for each sheet will be extracted as long as the layout name does not begin with “Layout”.

EXTRACT PROGRAM

Pulldown: AutoSD > Extract > Extract Program
Ribbon: AutoSD I > Shop bill



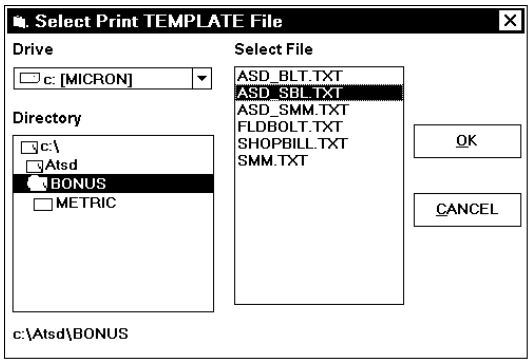
Use this command to run the extract program manually to convert an extracted bolt list or shop bill "txt" file, merge "imp", "kss", “csv” or "sbl" files or print "sbl" files. When extracting attributes manually you should choose "Space Delimited". Use the template file ASD_SBL.TXT for shop bills and ASD_BLT.TXT for

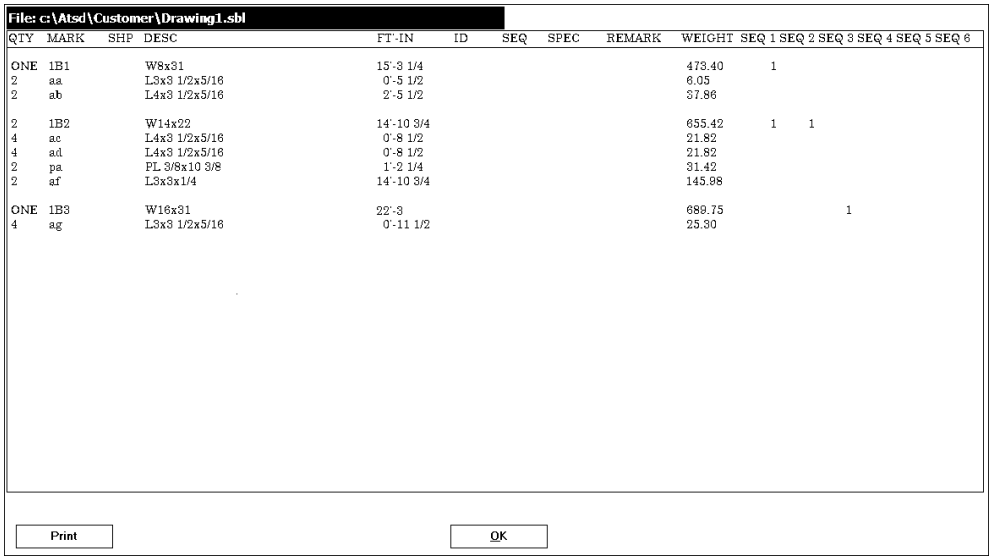
field bolts. These are located in your AUTOSD\BONUS folder. Or use a template file that you have created from these files.

The default folder for the template file is AUTOSD\BONUS. If you decide to modify the template file or create your own, copy it to your customer job folder before you modify it and access it there.
You should include the line BL:Y N005000 in the template file so the extract program will sort the lines so that the file will be in the same order as they appear in the drawing. After converting, the file name extension is changed to SBL for shop bills and BLT for field bolts.

You can choose to print the shop bill file when it is extracted or print it later. You have to have the BOLT LIST program that is sold with the CALCULATOR programs in order to print the field bolt files.

To print the converted file to the screen or to your printer select the template file folder and the template file used to extract the file and then pick OK. Pick the “Select File” button and change to the folder where your file is that you want to print and then pick OK. After you have selected the template file and the shop bill file pick the OK button at the bottom of the screen. A new screen pops up that displays the .sbl file. Pick the print button to send the file to your system printer.



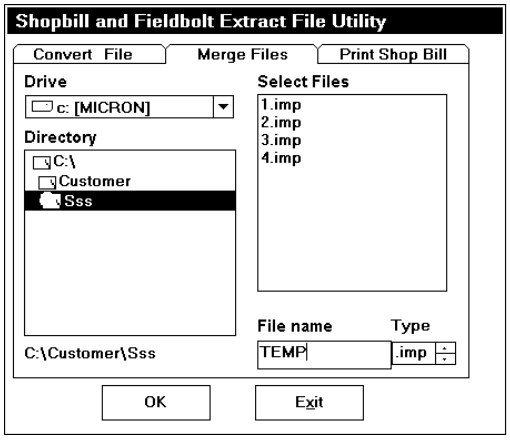


This is a sample print of a sbl file. If you were to print this file it would run off the right side of an 8 1/2" wide page because the lines are too long. This sample shows all of the columns that are generated from the asd_sbl.txt file. To make this print on the page without running off the edge you would need to edit the asd_sbl.txt file and remove the lines for the columns you do not want. The template file used to extract the attributes MUST be the same file used to convert or print the sbl file.

EJE
ATT

EXTRACT for EJE using Attributes

Pulldown: AutoSD > Extract > for EJE using Attributes
Ribbon: AutoSD I > Shop bill



Warning! Attributes must not be used if you are drawing in Model space and Paper space or just Paper space with multiple layouts.

AutoSD supports Structural Material Manger sold by E.J.E. Industries, Inc. P.O. Box 268, Washington, PA 15301, Phone: 1-(800) 321-3955, Fax: (412) 228-7668. You can order a full working demo from E.J.E. Industries.

When you select AutoSD > Extract > for SMM using Attributes from the pulldown menu, shop bill attributes will be extracted from the drawing and automatically converted to an ASCII file format that can be imported by the SMM program. The file name will be the same as the drawing name but will have an ".imp" or “csv” extension and will be located in the same folder as the drawing. The extension type is set in your customer configuration. “imp” is for the older fixed field format and “csv” is for the newer ASCII delimited file format.

After extracting for SMM you will have a separate “imp” or “csv” file for each drawing. You can merge these files together by using the extract program. Select Extract Program from the AutoSD > Extract pulldown menu. Select the tab “Merge Files”. Change the drive and then the folder to where the "imp" or “csv” files are located. Change the Type to ."imp" or “csv” Enter a file name in the “File Name” box. Do not give an extension. If the file does not exist it will be created to hold the merged files. If it does exist the files selected will be merged to it.

Select the files that you want to merge and select OK.

You can merge files into an existing file or create a new file to merge all files into. You cannot merge a file onto itself where new file and merging file have the same name. Extract for SMM uses some support files located in the AUTOSD\BONUS folder. They are asd_smm.txt and smm.txt. Do NOT modify these files or the program will not work.

Extract for SMM also works with shop bills created with older versions of AUTOSD without having to change the attribute block.

The description of a main member is usually the member size but sometimes you will need to use a description such as FRAME or BRACE for the main member. When such a description is given the length should be left blank. A main mark with no length will be treated differently than a main mark with a length.

EJE TXT

EXTRACT for EJE using Text

Pulldown: AutoSD > Extract > for EJE Using Text
Ribbon: AutoSD I > Shop bill

This program will extract the information when text is used in the shop bill. The program uses the shop bill configuration file to locate the columns of text, therefore, you must select the customer configuration file that was used to place the text in the shop bill.

The text must be lined up in columns and rows with the following restrictions.

Columns: The insertion points of the text in a column must line up within 1" on either side of the item as defined in the customer configuration file for the location of that column.

Rows: The insertion points of the text in a row must line up within 1" of each other.

Justification: The justification of all of the text in a single column must be the same but may be different from column to column.

Mark & Quantity: (1) If the main mark and sub mark are in the same column, their quantities must be in different columns. (2) If the main quantity and sub quantity are in the same column, their marks must be in different columns.

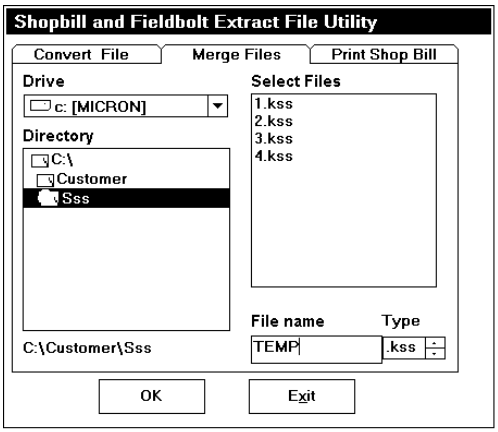
Any text that was selected whose insertion point does not line up with the column and row requirements will be omitted from the "imp" or “csv” file and written to a file with the same name as the "imp" or “csv” file but with an "err" extension. This could be weight values or text in columns not requested. In most cases these files may be deleted.

The text in the shop bill for Model space and each Paper space layout will be extracted as long as the Paper space layout name does not begin with “Layout”. All output will be in one file with the same name as the drawing file.

FAB ATT

EXTRACT for Fabtrol using Attributes

Pulldown: AutoSD > Extract > for Fabtrol Using Attributes
Ribbon: AutoSD I > Shop bill



Warning! Attributes must not be used if you are drawing in Model space and Paper space or just Paper space with multiple layouts.

AutoSD supports the KISS file format for material management software. When you select Extract for KISS using attributes, shop bill attributes will be extracted from the drawing and automatically converted to an ASCII file format that can be imported by the FabTrol program. If you used the tag

names ASD_NA for Project Name, ASD_NO for Job Number, ASD_GR for material grade and ASD_RE for the revision mark in your title block attributes, this information will be included in the ".kss" file. If you prefix or suffix the description with MI and the length is 0, MI will be used as the shape designation for miscellaneous material. Example: MI FRAME or FRAME MI.

Order of priority for grade: The grade for each piece is retrieved from the customer's Detail variables found under Beams. Entering a grade in the title block for ASD_GR will override the customer's Detail variables setting for all grades. If the grade is listed in the BOM for a piece, that grade will override all other settings.

The file name will be the same as the drawing name but will have an ".kss" extension and will be located in the same folder as the drawing. After extracting for KISS you will have a separate "kss" file for each drawing. You can merge these files together by using the extract program. Select Extract program from the AutoSD > Extract pulldown menu. Select the tab "Merge Files". Change the drive and then the folder to where the kss files are located. Change the Type to kss. Enter a file name in the "File Name" box. Do not give an extension. If the file does not exist it will be created to hold the merged files. If it does exist the files selected will be merged to it.

Select the files that you want to merge and select OK.
You can merge files into an existing file or create a new file to merge all files into. You cannot merge a file onto itself where new file and merging file have the same name.

EXTRACT for KISS uses some support files located in the AUTOSD\BONUS folder. They are kiss.lsp and asd_kiss.txt. Do NOT modify asd_kiss.txt or the program will not work.

EXTRACT for Fabtrol using Text

Pulldown: AutoSD > Extract > for Fabtrol Using Text
Ribbon: AutoSD I > Shop bill

This program will extract the information when text is used in the shop bill. The program uses the shop bill configuration file to locate the columns of text, therefore, you must select the customer configuration file that was used to place the text in the shop bill.
The text must be lined up in columns and rows with the following restrictions.
Columns: The insertion points of the text in a column must line up within 1" on either side of the item picked to define the location of that column.
Rows: The insertion points of the text in a row must line up within 1" of each other.
Justification: The justification of all of the text in a single column must be the same but may be different from column to column.
Mark & Quantity: (1) If the main mark and sub mark are in the same column, their quantities must be in different columns. (2) If the main quantity and sub quantity are in the same column, their marks must be in different columns.

After the customer is selected the text is processed. Any text that does not line up with the column and row requirements will be omitted from the "kss" file and written to a file with the same name as the "kss" file but with an "err" extension. This could be weight values or text in columns not requested. In most cases these files may be deleted.

If you used the tag names ASD_NA for Project Name, ASD_NO for Job Number, ASD_GR for material grade and ASD_RE for the revision mark in your title block attributes, this information will be included in file ".kss".
Order of priority for grade: The grade for each piece is retrieved from the customer's Detail variables found under Beams. Entering a grade in the title block for ASD_GR will override the customer's Detail variables setting for all grades. If the grade is listed in the BOM for a piece, that grade will override all other settings.

The text in the shop bill for Model space and each Paper space layout will be extracted as long as the Paper space layout name does not begin with "Layout". All output will be in one file with the same name as the drawing file.

FabSuite *.kss file

The file "C:\autosd\bonus\FabSuite Shape List.pdf" is a list of Key words the Extract for KISS program will recognize when creating the *.kss files for FabSuite. You must use the text from the BOM TEXT column of this file in your description in your bill of materials in order to export that material to the *.kss file. Your material description may be upper or lower case.

COMPILE QUANTITIES From ALL DRAWINGS

Pulldown: AutoSD > Compile Qtys from All drawings using attributes
Pulldown: AutoSD > Compile Qtys from All drawings using text
Ribbon: AutoSD I > Shop bill

Warning! Attributes must not be used if you are drawing in Model space and Paper space or just Paper space with multiple layouts.

Use this command to extract the shop bill using attributes or text from all drawings in one folder. If more than one drawing is in the selected folder each drawing will be opened one at a time and the bill of material extracted. The text in the shop bill for Model space and each Paper space layout will be extracted as long as the Paper space layout name does not begin with "Layout". After all drawings have been processed the extracted files are merged into one file and converted. The converted file name is "bom_qty.txt" and will be located in the folder where the drawings are that were processed. The information written to this file is the Mark, Quantity, Description, Length and Grade. You have the option of combining all common marks and sorting like material together. Sorting will list all "W" shapes together, all "L" shapes together, etc. In order to sort by shape, the shape designation must be at the left end of the description. Shapes recognized to sort are as follows: HP, M, S, W, C, MC, MT, ST, WT, HSS, TS, HSS, PS, L, CF, FLAT BAR, BAR, UM PL, FB, PL, UM PLATE, PLATE, TUBE,

SCH 40, SCH 80, SCH 120, XX-STRG, X-STRG, STD PIPE, PIPE, RAIL, A325, A307, A490, BOLT, ROD, REBAR, EXPANDED and GRAT as well as the prefix you set in the configuration for plates under FLAT BARS / PLATES.

EXTENDING WEIGHT

If the weight category is not 0 the weight for each piece will be calculated and written in the weight column. The weight is the total weight for the quantity of each piece listed. The total for the piece shipped is not given. You can use the command AutoSD > Calculator > Select Dim TO Add > Decimal from the pulldown menu to quickly get a total if needed. Listed below are the valid descriptions that the weight program will recognize.

KEY WORDS	DESCRIPTION EXAMPLES				
A307	`W`O A307	or	`W`O A307 BOLT		
A325	`W`O A325	or	`W`O A325 BOLT		
A490	`W`O A490	or	`W`O A490 BOLT		
BAR	BAR `W x 1`8	or	BAR `W`[	or	BAR `W`{
BOLT	`W`O BOLT				
C	C8x11.5				
CLEVIS	# 2`8 CLEVIS	or	CLEVIS # 2`8		
EXP	4.0# EXP METAL				
EXPANDED	4.0# EXPANDED METAL				
FB	FB `4 x 2				
FLAT BAR	FLAT BAR `4 x 2				
GRAT	GRAT `3 x 1`4 x 36				
HP	HP13 x 87				
HSS	HSS 6.625 x .5	(round pipe)			
HSS	HSS 6.625 x `8	(round pipe)			
HSS	HSS 6 x 2 x `3	(rectangular tube)			
L	L4 x 3`8 x `5				
M	M8 x 6.5				
MC	MC12 x 20.7				
MT	MT3 x 10				
`P	`P`6 x 13`4	or	`P 14 GA x 13`4		
PIPE	1`8`O STD	or	1`8`O PIPE	or	1`8`O STD PIPE
PLATE	PLATE `6 x 13`4	or	PLATE 14 GA x 13`4		
PS	PS1`8`O		(This will give the weight for std pipe)		
RAIL	60# RAIL		(Do not use this key word for handrail pipe)		
REBAR	#8 REBAR	or	REBAR #8		
ROD	`W`O ROD				
S	S12 x 50				
ST	ST6 x 15.9				
STD PIPE	1`8`O STD PIPE				
SCH 40	1`8`O SCH 40	or	1`8`O SCH 40 PIPE	(Same as STD PIPE)	
SCH 80	1`8`O SCH 80	or	1`8`O SCH 80 PIPE	(Same as X-STRG PIPE)	

SCH 120	1`8`O SCH 120	or	1`8`O SCH 120 PIPE (Same as XX-STRG PIPE)
TS, TUBE	TS 3 x 2 x `4	or	TUBE 3`8x3`8x`4
TURNBUCKLE	TURNBUCKLE `W x 6		
UM `P	UM `P`6 x 13`4		
UM PLATE	UM PLATE `6 x 13`4		
W	W14 x 22		
WT	WT12 x 27.5		
X-STRG	1`8`O X-STRG	or	1`8`O X-STRG PIPE
XX-STRG	1`8`O XX-STRG	or	1`8`O XX-STRG PIPE

Key Words By Country

USA	CANADA	UK	EUROPE	AUSTRALIA
C	C	BT	DIL	BT
CF	CF	C	HD	CT
HP	DN	CT	HE	CHS
HSS	HP	CHS	HL	EA
L	HSS	ERW	HP	PFC
M	L	J	HSS	RHS
MC	M	L	HX	TFB
MT	MC	UB	IPE	TFC
PS	S	UC	IPEA	UA
S	W	UBP	IPEV	UB
ST	MT	RHS	IPN	UC
TS	ST	SHS	L	WB
W	WT	U	WC	
WT	TS	UAP		
	UA	UPN		
	WWF			
	WWT			
	WRF			

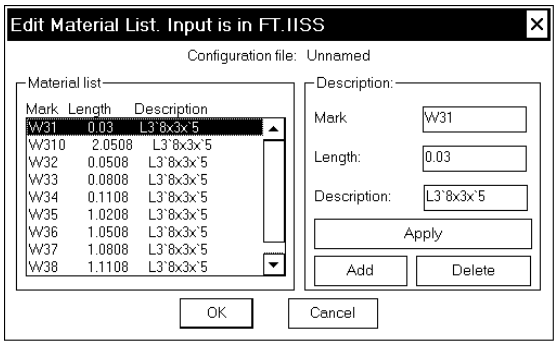
In the above examples the KEY WORD may be placed at the beginning or the end of the description. Only Australian key words may be placed in the middle, (150UB 14). You may use upper or lower case letters. The dimensions for Angles, Tubes, Bars, Flat Bars and Plates may be in any order. The first two dimensions for GRATING may be reversed but the width must be the third dimension in the description. Spaces are permitted anywhere in the descriptions but not in the key words except where shown. In all cases the descriptions must use the custom text supplied by AutoSD using the backward apostrophe (`) or tilde (~) characters for fractions. If a description is not recognized no weight will be given. Grating is for welded steel grating only. Cross bar centers can be 2" or 4" and bearing bar centers can be 15/16, 1 3/16, 1 3/8, 1 7/8, or 2 3/8.

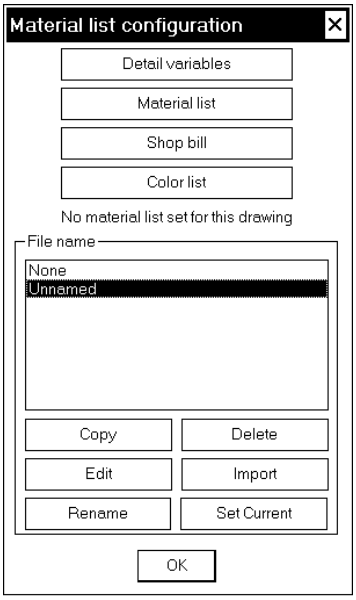
The weight calculated is for non-galvanized carbon steel. When listing common pieces it is customary to describe the piece completely the first time and then give just the quantity and mark of the piece each time after that. Each time you describe a new piece it is added to a list

that is held in memory. When you give only the quantity and mark of a piece the program searches this list for the same mark. If it finds it, the weight will be extended for the common mark. When you end the drawing this list is erased from memory. This is why the shop bill should be filled out completely in one drawing session in order for all weights to be extended.

See chapter 5 for the manual EXTEND WEIGHT program.

MATERIAL LIST





This file must be modified before it can be used. A sample file is supplied called “Unnamed” and is shown to the right. To edit the file select AutoSD > Customer configuration and pick “Material list”. Select “Unnamed” and pick “Edit”. You can change the mark, length and description. You can add and delete marks. Each time a sub assembly mark is entered when using the shop bill program, the program searches this file for the mark. If it is found, the length and description are read from the file and automatically entered in the shop bill. If it is not found or you selected “None” for the material list file you must enter the length and description manually. Use this file to list all standard marks for clips, plates, bolts, etc.

Note: Marks and descriptions for standard end connection angles for beams are saved in the customer configuration and do not need to be added to this file.

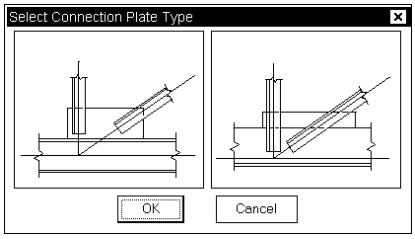
If you had an older version that used the material.tbl file you can import it with the Import button. You can also import individual configurations from other custMAT.cfg files.

Chapter 11 - Trusses

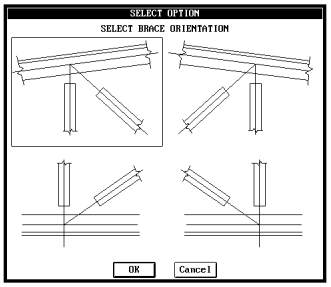
Pulldown: AutoSD > Bracing > Calculate > Gusset PL (Truss)
Toolbar: Bracing
Ribbon: AutoSD II > Bracing

The TRUSS portion of the BRACE program is for calculating the size of the gusset plate, the length of weld and clearance dimensions at a panel of a welded truss. The top and bottom chords can be either wide flange shapes, angles or split tees. The truss program is supported by icon menus to select the type of truss, top or bottom chord and orientation. A full screen view of the type selected is provided for your reference in entering the necessary data.

`Command:` Pick [AutoSD or AutoSD II] > Bracing > Calculate Gusset PL (Truss)



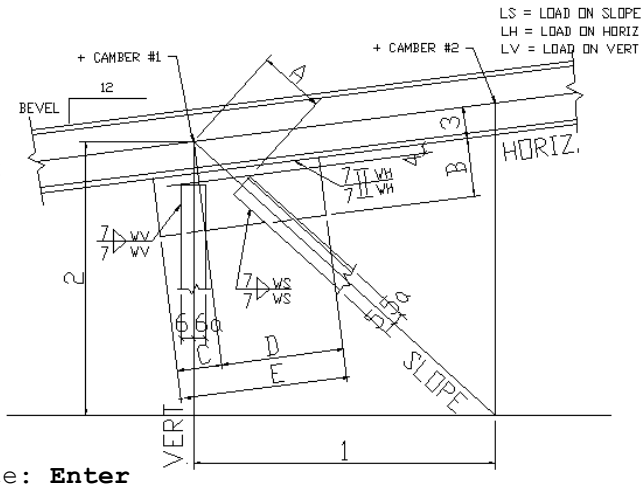
An icon menu appears
Pick the truss type



Another icon menu appears.
Pick the chord and orientation.

A slide drawing is put on the screen of the truss type, chord and orientation that you picked.

```
Command:
(DECIMAL) Load on slope member (kip): 55
(DECIMAL) Area of slope member(s): 4
(DECIMAL) Load on vert. member: 80
(DECIMAL) Area of vert. member(s): 8
(1)Single or (2)Double angles (1/2) <1>: 2
(FT.IISS) Input #1: 8.06
(FT.IISS) Input #2: 6.03
(FT.IISS) Camber #1: .0008
(FT.IISS) Camber #2: .0010
(FT.IISS) Input Bevel: .02
(FT.IISS) Input #3: .04
(FT.IISS) Min clearance #4 <1/2">: Enter
(FT.IISS) Input "A" or <ENTER> to calculate: Enter
(FT.IISS) Input #5: .0108
(FT.IISS) Input #5a <1 1/2">: Enter
(FT.IISS) Input #6: .02
(FT.IISS) Input #6a <2">: Enter
```



```
(INTEGER) Input weld size in 16ths: #7 <3>: Enter
Command: REDRAW                                          Finished
```

At the prompt *(FT.IISS) Input "A" or <ENTER> to calculate:* you can input a dimension if you are checking a drawing made by someone else or you can press the enter key. If you input a dimension the program will check to see if it works with the given clearance. If it does, fine. if not, the correct dimension for "A" will be calculated and used. Either way, the answer for "A" that is displayed in the output will be the one that works.

Chapter 12 - Stairs

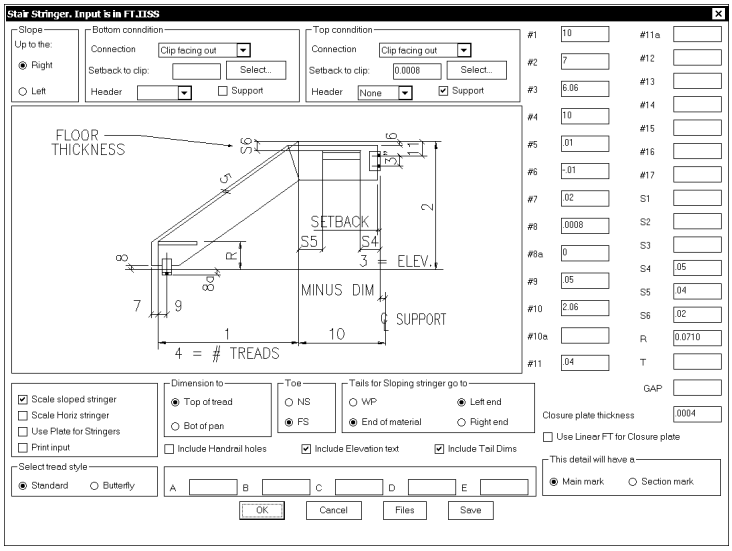
All programs in this chapter are under the AutoSD > Stair pulldown menu, Stairs / Rails Toolbar and AutoSD II > Stairs ribbon menu unless noted otherwise.

The STAIR program uses icon menus and dialog boxes for user friendly input. You can draw pan stairs, bolted tread stairs and "butterfly" stairs for pre-cast treads that sit on top of the stringer. The orientation of the stringer can be sloping up to the right or sloping up to the left. Tail dimensions for bolted treads will be determined for you and put in a easy to read table or on the stringer detail if you select to draw the length to scale. There are many different pan options to choose from and you can create your own to be used by the stringer program with the easy to use Tread Builder. There are four top and seven bottom stringer conditions to choose from. You can have holes for handrail post automatically located and placed on the stringer.

The stair stringer can be drawn to the actual length and to the actual bevel. The depth will be drawn to scale.

Your sheet should be set up to a scale of 1" = 1'-0, (1:12 for metric)

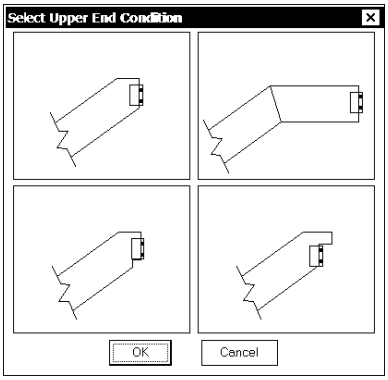
`Command:` Pick [AutoSD or AutoSD II] > Stairs > Stringer



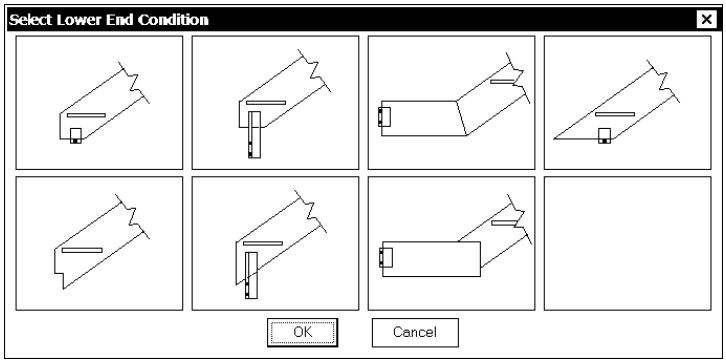
A dialog box is used for input showing the top and bottom conditions and all of the input from the previous stringer. The end conditions are changed by picking the options under “Connection at bottom” and “Connection at top”. Under “Select tread style” "Butterfly" is for pre-cast treads that sit on top of the stringer. These treads will be supported by flat bars.

Use the tab key to move from one box to the next. Use shift + tab to move up to the preceding box.

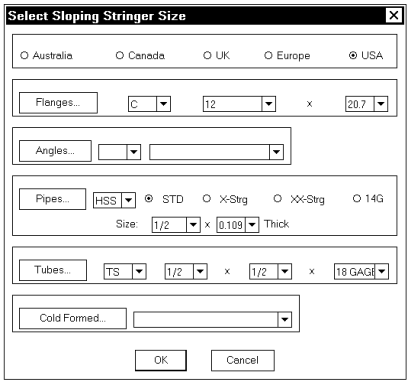
If the stair has 6 treads or less, the length will be drawn to scale otherwise you would need to check the box for "Draw Length to Scale" if you want to draw the stringer length to scale.
If you are saving CNC data and will have bolted bracing under the stair, the far side stringer will need tails from the left end and the near side stringer from the right end of material.



An icon menu is used when you click on “Select” under “Connection at top”



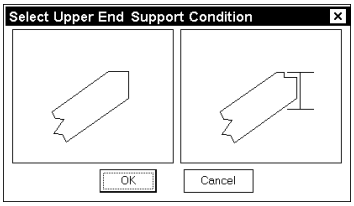
An icon menu is used when you click on “Select” under “Connection at bottom”



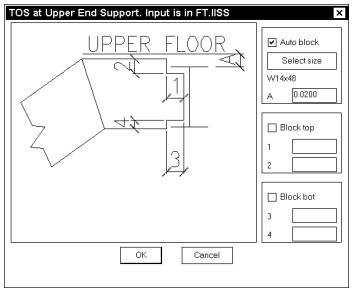
After entering all of the required dimensions in the box above and picking OK, a dialog box is displayed to select the stringer sizes. Select the shape, depth and weight and then select OK.

Insertion point: A box is displayed around the cross hairs outlining the area required for the stair detail.

```
              Main mark: 1S1
Are marks RIGHT and LEFT (Yes/No) <N>: Enter
         Quantity: 1
Shop bill information on mark 1S1
Select/Place shop bill text on LINE <4>: Enter
```



An icon menu is displayed to select whether to apply blocks or not to the upper end. The left selection is for an end with no blocks in the top or bottom flange. You still get to enter the “minus” dimension. The right icon is selected to enter blocks.



A dialog box is used to enter the blocks. If you select “Auto block” you select a beam size and enter dimension “A” for the offset from the floor elevation to the TOS of the support beam. The blocks required are calculated for you. If you select “Block top” or “Block bot” you enter the width and depth of the blocks from end of material.

(FT.IISS) Horiz. Minus dimension at upper end <-1/8">: **Enter**
Sub assembly mark for sloping stringer: **mb**
Sub assembly mark for clip at lower end: **aa**
Clip is (1)NS, (2)FS, (3)BS <1>: **Enter**
(FT.IISS) Gage of clip: **.0112**

Select angle from dialog box

Add piece to shop bill (Yes/No) <Y>: **Enter**
Shop bill information on mark aa
Total sub quantity: **1**
Bolt size: (5)5/8" (6)3/4" (7)7/8" (8)1" <6>: **Enter**
Number of field bolts in Bottom end <1>: **Enter**
(FT.IISS) Total grip less washers <5/16">: **.0112**
Add another field bolt (Yes/No) <N>: **Enter**
Sub assembly mark for closure plate at lower end: **pa**
Add piece to shop bill (Yes/No) <Y>: **Enter**
Shop bill information on mark pa
Total sub quantity: **1**
Sub assembly mark for upper stringer: **ma**
Sub assembly mark for clip at upper end: **ab**
Clip is (1)NS, (2)FS, (3)BS <1>: **Enter**

Separate spacing of holes with a comma

(FT.IISS) Gage of clip <1 3/4">: **Enter** Enter gage as .02,.0112 for 1 row, 2 columns

Select angle from dialog box

Add piece to shop bill (Yes/No) <Y>: **Enter**
Shop bill information on mark ab
Total sub quantity: **1**
Bolt size: (5)5/8" (6)3/4" (7)7/8" (8)1" <6>: **Enter**
Number of field bolts in Top end <2>: **Enter**
(FT.IISS) Total grip less washers <5/16">: **.0010**
Add another field bolt (Yes/No) <N>: **Enter**
Sub assembly mark for floor support angle at upper end: **ac**
Floor support angle is (1) NS, (2) FS <1>: **Enter**

Select floor support angle from dialog box

Add piece to shop bill (Yes/No) <Y>: **Enter**
Shop bill information on mark ac
Total sub quantity: **1**

The Tread builder dialog box is displayed. Select 10GA 2-2-6-Angle 1 1/2

Sub assembly mark for vertical support angles at top riser: **ad**
Add piece to shop bill (Yes/No) <Y>: **Enter**
Shop bill information on mark ad
Sub assembly mark for horizontal support angles at top riser: **af**
Add piece to shop bill (Yes/No) <Y>: **Enter**
Shop bill information on mark af
Sub assembly mark for horizontal support angles at standard pans: **ag**
Add piece to shop bill (Yes/No) <Y>: **Enter**

```
Shop bill information on mark ag
Pick a point in "clear" area for blowup of pan:
Select dimension line locations:
Quantity of this stair <1>:
Sub assembly mark for Standard pan: ha
```

The Tread builder dialog box is displayed. Select 10GA 2-2-1-Angle 1 1/2

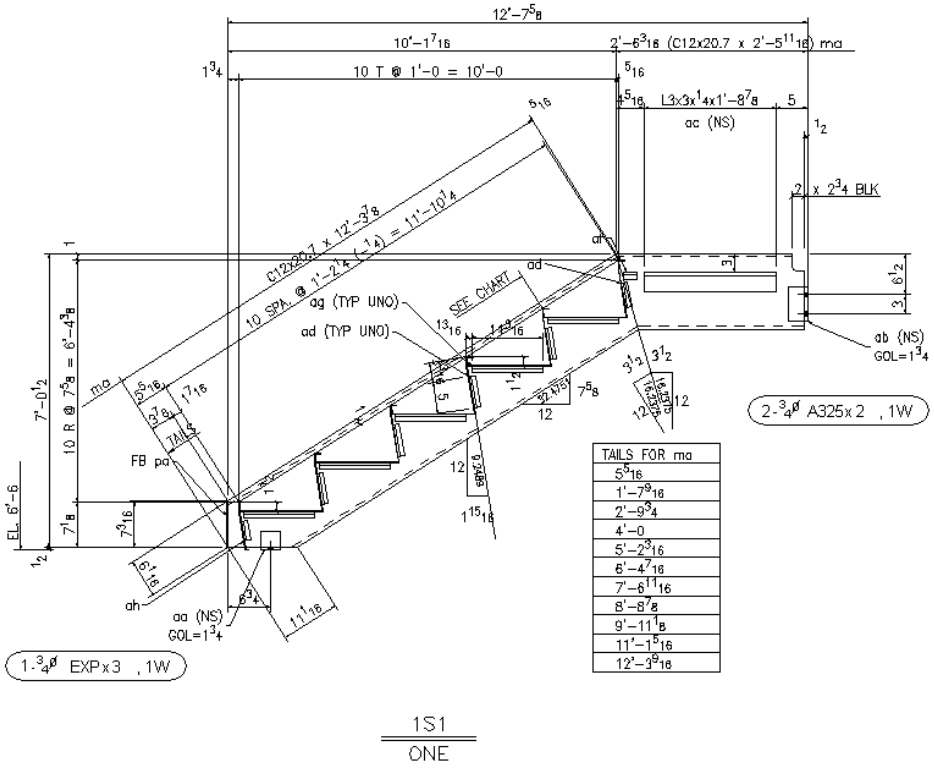
```
Sub assembly mark for vertical support angles at bottom riser: ah
Add piece to shop bill (Yes/No) <Y>: Enter
Pick a point in "clear" area for blowup of pan:
Select dimension line locations:
Quantity of this stair <1>:
Sub assembly mark for Standard pan: hb
Shop bill information on mark hb
Add upper stringer mark ma to shop bill (Yes/No) <Y>: Enter
Shop bill information on mark ma
     Total sub quantity: 1
Add sloping stringer mark mb to shop bill (Yes/No) <Y>: Enter
Shop bill information on mark mb
     Total sub quantity: 1
Command:
```

Finished.

- If the stringer had been drawn going up to the left the tail dimensions would be from the upper end. Either way you draw the stair the tails will be from the end selected in the main dialog box.
- If you checked the box to draw the stringer length to scale the tail dimensions for the treads will be put on the detail instead of in a chart.
- If you are saving CNC data and the stair length is not to scale the "tail chart" will be used by default. The CNC xdata will be associated with the tail text instead of hole entities. If you need to change the tail dimensions in the chart you must use the *Replace Dimension* command instead of a text editor if you saved CNC data so the xdata will be changed.
- If you would like additional dimensions on the stair you can use the dimension command and use the defaults given since the end of the stair is drawn to scale. The stair is also drawn to the exact bevel.
- Blocks are dimensioned to end of member or to clip as set in your Beam configuration.
- Block length is based on "Clear dim added for block length" as set in your Beam configuration. The Beam Configuration can be set even if you did not purchase the Beam and Column package.
- Clips and plates for header connections, clips at all upper end stringer connections and clips at lower end of horizontal stringer are, by default, 1 column and 2 rows. You can make them 2 columns and 1 row by entering the two gages separated by a comma when you see this prompt. Separate spacing of holes with a comma. (FT.IISS) Gage of clip: .02,.0112
- If you check the box "Use Linear FT for Closure plates, one mark will be used for all closure plates and the total length will be rounded up to the nearest foot.
- **Files**: Selecting "Files" opens a dialog box that allows you to save to, delete or load a stringer from the stair.lst file that is in your customer's job folder. This is where you load the values for stringers that were exported from the Elevation Generator program. Some values, such as the location of holes, are not carried over from the Elevation Generator so they need to be checked. Once a stringer

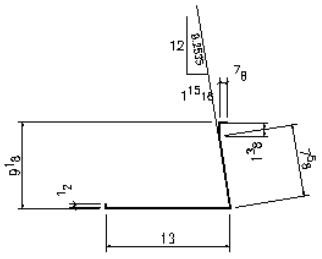
is detailed, all values become the default for the next stringer. Saving a stringer here allows you to recall the input when detailing the rail or Erection elevation view or another similar stringer.

Note:
See appendix E for forms that can be filled out by your more experienced detailer and given to a less experienced person for entering information into the stair program.

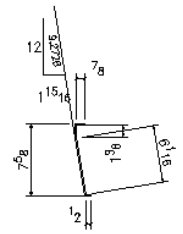


TAILS FOR ma
5 5/16
1'-7 9/16
2'-9 3/4
4'-0
5'-2 3/16
6'-4 7/16
7'-6 11/16
8'-8 7/8
9'-11 1/8
11'-1 5/16
12'-3 9/16

1S1
ONE

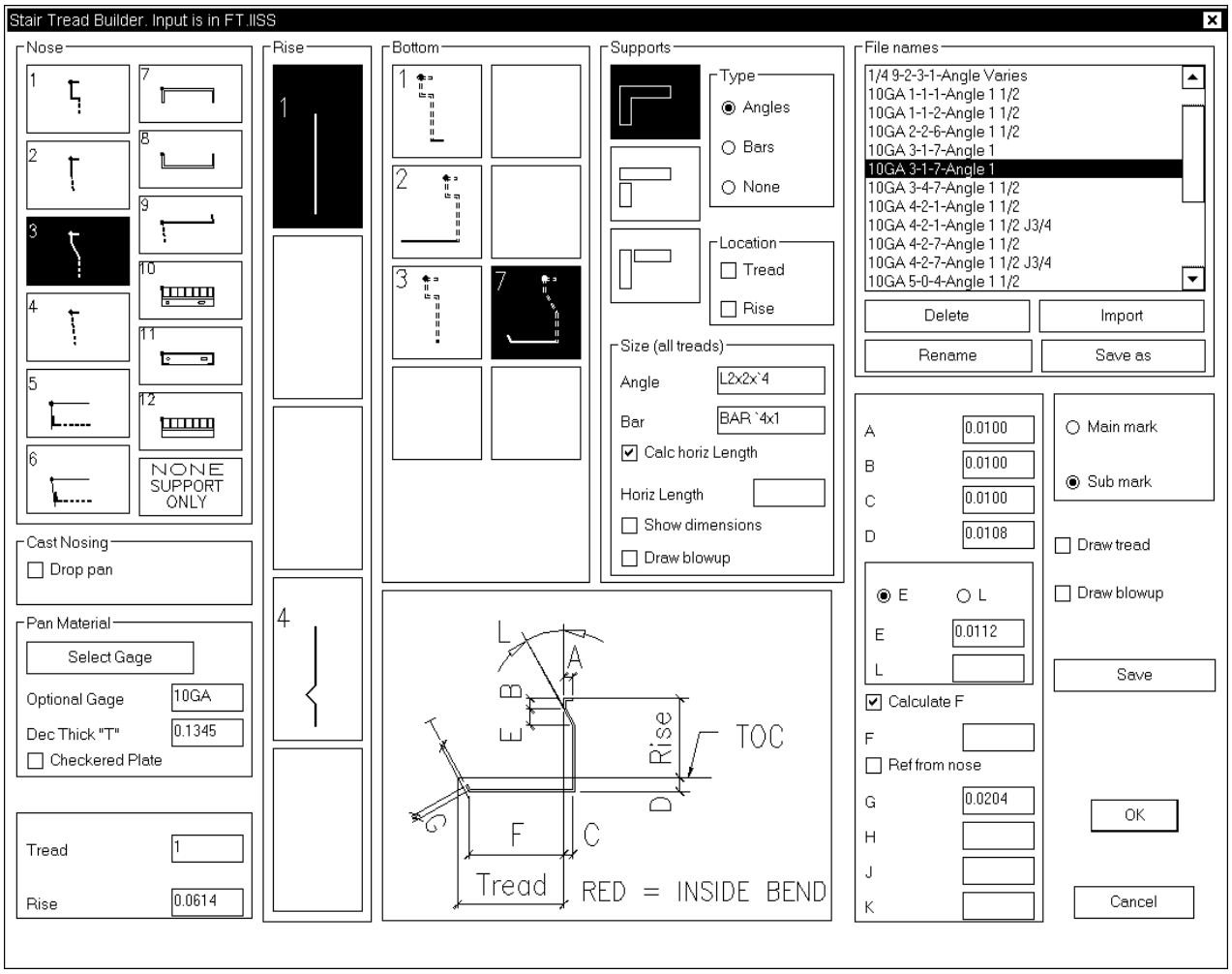


DETAIL OF STANDARD PAN
PL10GAx7 5/8(±1 1/16) x 1'-11 5/16 (BENT)– ha
(10 REQ'D)



DETAIL OF BOTTOM PAN
PL10GAx7 5/8(±1 1/16) x 8 13/16 (BENT)– hb
(1 REQ'D)

CREATING PAN TREADS with the Tread Builder



1. Select the type of nose. Selecting “None. Support Only” will just draw the supports on the stringer.
2. Select the type of rise available for the nose that was selected.
3. Select the type of bottom available for the nose and rise that was selected.
4. Select the Support configuration. The top option is for mitered supports.
5. Select the support type. Putting a check in the box for “None” will cause supports to NOT be drawn under the tread and rise.
6. Put a check in the box for “Tread” if you want a support under the tread.
7. Put a check in the box for “Rise” if you want a support on the riser
8. Put a check in the box for Drop pan if you have a cast nosing
9. Pick the “Select Gage” button if a gage material is needed. If a gage is entered it will be used in the description, (optional). Leave the gage box blank to use the thickness in the description.
10. Enter a thickness for the pan material, (required)
11. Put a check in the box for Checkered Plate if it is Checkered Plate.
12. Enter a value for Tread, Rise and A - K as required.

Putting a check in the box for “Calculate F” will let the program calculate the tread width from the value entered for Tread. Removing the check will force the program to use the value “F” for the tread width.

Delete: This will delete the selected tread from the File names list.
Import: This will allow you to import a tread from another TRDBILDR.CFG file
Rename: This will allow you to rename a file name.
Save as: This will save an existing tread as a different file name, making a copy of it.
Save: This will save the current file selected.

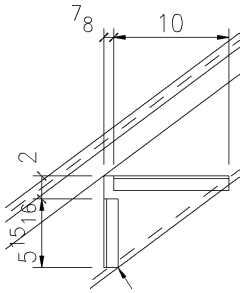
Size (all treads):
Values here are global, one value for all treads. All other values are saved for each tread.
Enter the Angle and Bar size using the custom text character for fractions.
Putting a check in the box for “Calc horiz length” will cause the horizontal support length to be calculated based on the tread width.
Putting a check in the box for “Show dimensions” will cause dimensions to be placed on one set of supports on the stringer.
If the top icon for Supports is selected and you putting a check in the box for “Draw blowup”, the program will draw a separate detail of the mitered support with dimensions at a larger scale.

Option E and L: For nose type 3.
Select “E” to enter dimension “E” for calculating angle “L”.
Select “L” to enter the angle “L” for calculating dimension “E”

Main mark: This option will put a main mark for the tread when drawing a Tread End View.
Sub mark: This option will put a sub mark for the tread when drawing a Tread End View.
Draw tread: Putting a check in this box will cause the tread to be drawn.
Draw blowup: Putting a check in this box will cause a larger end view detail to be drawn of the tread.

Pick OK to draw tread.

Notes:
If you need a pan riser to be vertical from nose to bottom use Nose 4, Rise 2, bottom 7 and set dimension “C” to 0
Nose type “NONE SUPPORT ONLY” is for easy placement of support angles or bars when the tread may be a concrete insert.
Input G can be 0 for nose 9 if "Ref from nose" is unchecked. This will eliminate the upward turn at the inside edge of the tread.



DIM PAN SUPPORTS

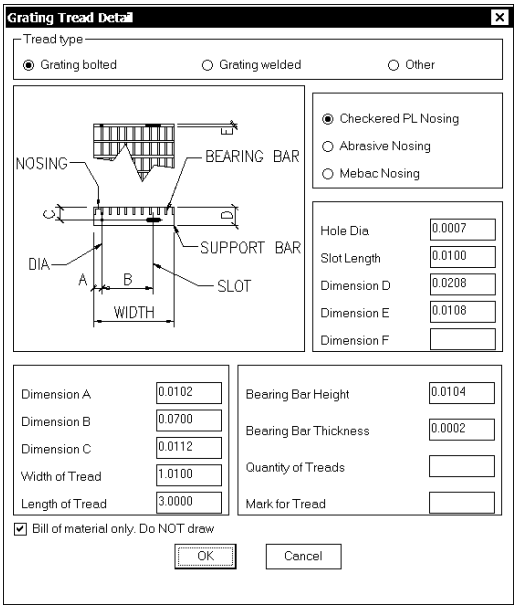
This command is used to add dimensions to existing pan supports. If the pans are drawn you will need to zoom in real close so you can pick the

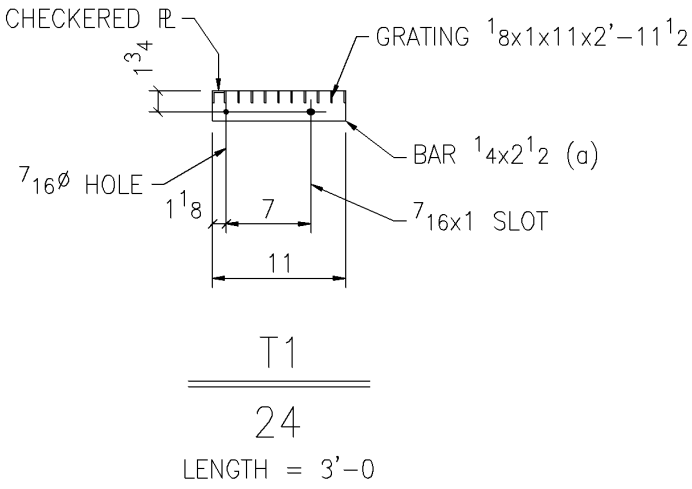
supports and not the pan. Your first selection is the top line of the horizontal support. Your second selection is line on the front side of the vertical support that is immediately below the horizontal support. After picking the vertical support the command pauses to let you zoom out. You must zoom out far enough so that all dimensions will be in the viewing area.

BOLTED TREAD END DETAIL

This draws the end view of a bolted or welded grating tread or a bolted bent plate tread, (other). The scale of the detail is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.

This box is also opened by the stair stringer program when you select a grating tread. You can pick "Bill of material only. Do NOT draw" or you may pick cancel to omit all output from this program without stopping the stair detail.

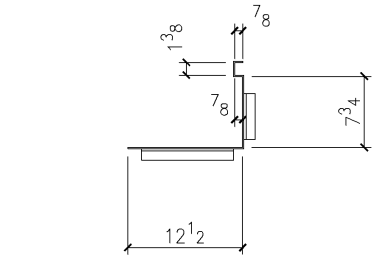





```
Insertion point at upper left corner of tread:
Piece mark for support bar
Enter mark or use <a>: Enter
Shop bill information on mark T1
Place shop bill text on LINE <16>: Enter
Shop bill information on mark T1
Place shop bill text on LINE <13>: Enter
```

PAN TREAD END VIEW

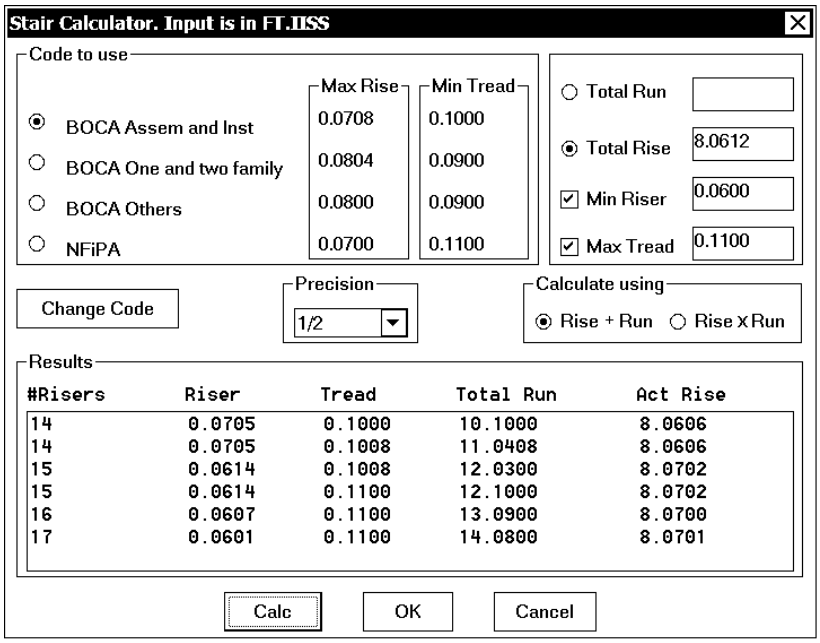
This uses the Tread Builder to select the pan and options. It draws the end view of a pan. If a blowup is drawn, only the pan will be drawn, otherwise, you can elect to show or not show supports and dimensions. If you choose to show dimensions you will be asked to pick the location of each



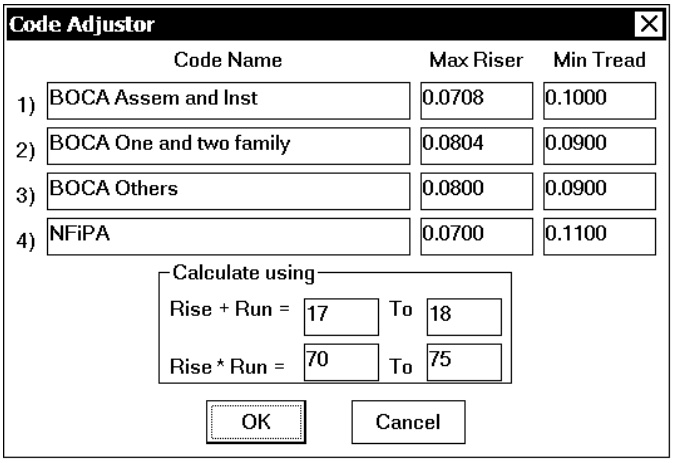
DETAIL OF STANDARD PAN
PL 1/8x23 3/8(±1/16) x 3'-0 (BENT)-ha
(1 REQ'D)

dimension. “Rubber band” lines for the extension lines will show you which dimension you are working with.

STAIR TREAD / RISER CALCULATOR



This is a design program used to calculate a tread/riser combination for a given total rise or total run. Four different codes are given to choose from. Each has their own standard maximum riser and minimum tread which can be changed by the user. Additionally, you can enter a minimum riser and maximum tread. The tread/riser combination is calculated using either "Rise + Run" or Rise x Run". The "Precision" box determines the increment of the tread width. The results are shown in the list box at the bottom.

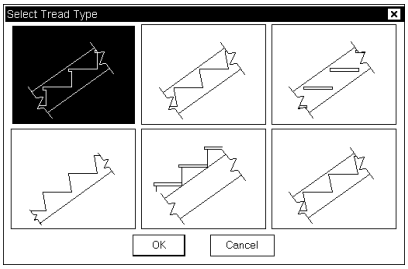


Pick "Change Code" and the dialog box shown to the left is displayed. Here you can change the values by picking the box you want to change and editing the contents. All boxes can be edited. Picking OK saves the changes and takes you back to the previous box. Picking Cancel does not save any changes.

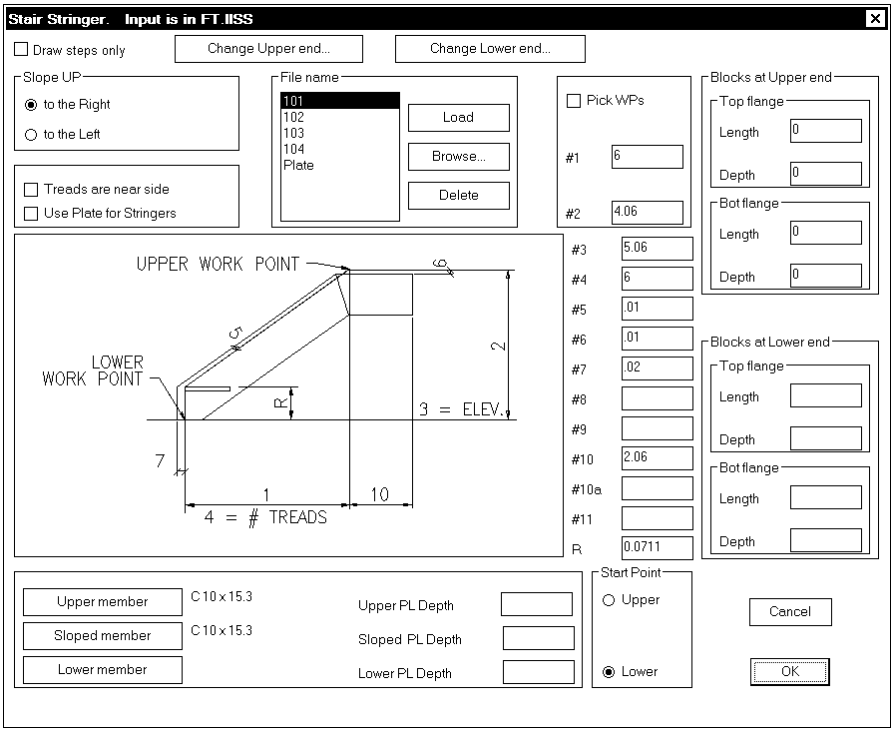
STAIR ELEVATION

This program is for drawing an elevation view of a stair stringer on an erection drawing. Input is with icon menus and dialog boxes.

`Command:` Select [AutoSD or AutoSD II] > Stairs > Erection > Stair Elevation



Select the tread type



Change Upper end:
Change Lower end:
Pick these to select an end condition from an icon menu.

Slope UP
Select “to the right” or “to the Left” to change the orientation of the stair.

Draw steps only: Checking this option will reduce the amount of input required and will draw the treads as solid lines but will not draw any other part of the stair or rail.

Treads are near side: Checking this option will draw the treads as solid lines and the flanges of a channel stringer as hidden. Leaving it unchecked will have the opposite effect.

Use Plate for **Stringer:** Checking this option will let you input the depth of the stringer and will not have any flanges. Leaving it unchecked will allow you to selecting a "C" or "TS" shape.

File name: This shows a list of names that were saved by the stair stringer program. Select a name and pick **Load** to set all of the inputs to match the stair that created the file. **Browse** will let you select a list from another folder. **Delete** will delete the file selected. If you exported stringers from the Elevation Generator you must open the file with the Stair Stringer program to convert the file before it can be used by the Stair Elevation program.

PICK WPs: Checking this option will disable boxes for dimension #1 & #2 and the program will ask you to pick the upper and lower points.

Start Point: You can choose to pick the lower work point or the upper work point as the starting point as shown in the slide picture in the dialog box above.

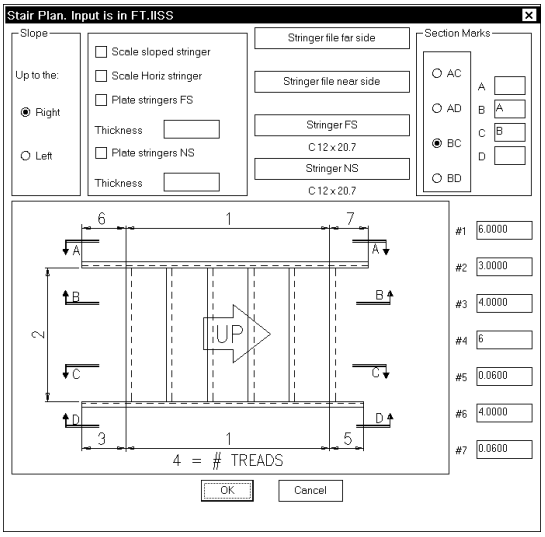
If a block is NOT required, enter 0 for the length and depth or leave these blank.

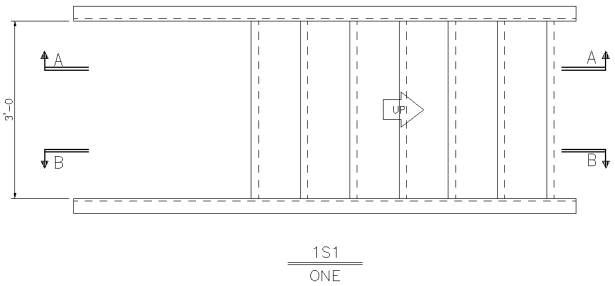
```
Pick Lower work point at finish floor:
Pick location of Horizontal dimension:
Pick location of overall Vertical dimension:
```

PLAN VIEW OF STAIR

Command: Pick [AutoSD or AutoSD II] > Stairs > Plan View of Stair

You can draw a plan view of a stair by picking "Plan View of Stair" from the pulldown menu. The plan can be to scale. If you saved the stringer file when detailing the stringer you pick the "Stringer file" buttons and have some information enter for you from the files.



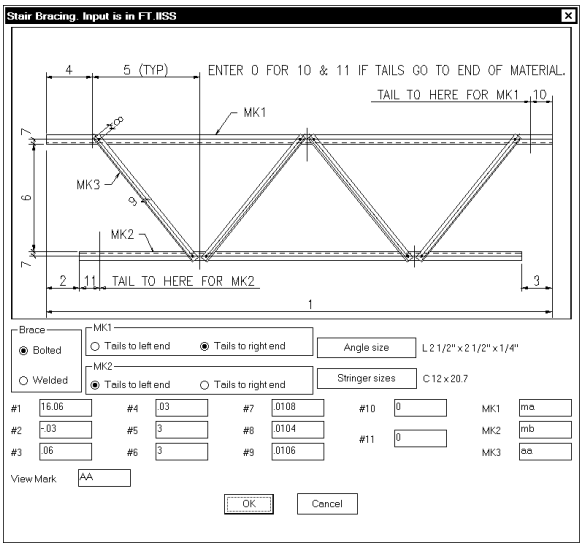


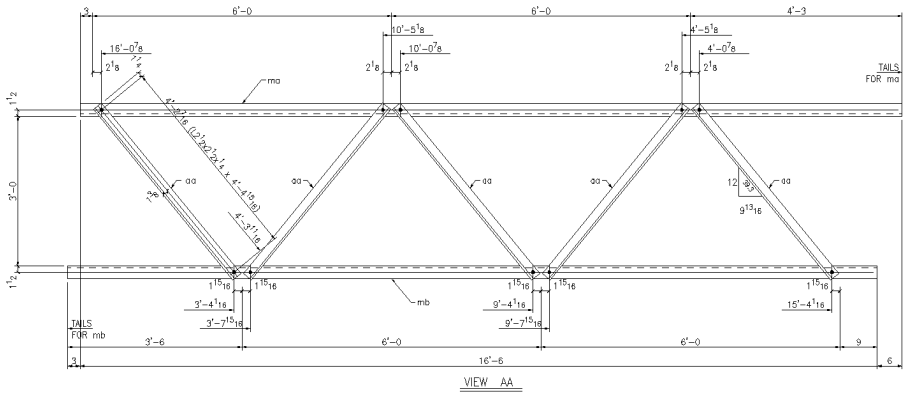
Section marks can be facing toward the back or toward the front. Picking "AC" will put section marks at "AA" and "CC" and you enter the mark you want to use at section "A" in the box labeled "A". Do the same for section "C". Enter values as required and pick OK to draw the plan.

STRINGER BRACING

This program will detail a view of the stair, looking up from the bottom, detailing shop bolted or shop welded bracing on the underside of the stair stingers. Tail dimensions may go to either end of material or to a work point.

Note:
When detailing the stringers and saving CNC data you will need to have the tails coming from the right end of one stringer and the left end of the other stringer to match the stringer details. Both stringers and stringer bracing will have to be on the same drawing.





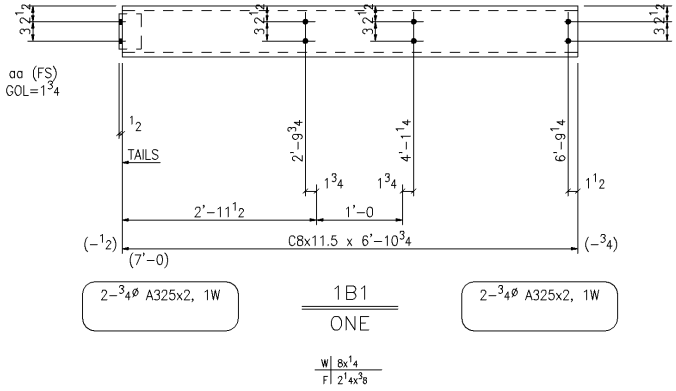
Header Beam

This program is for detailing simple header beams at stair landings. It is supplied with the Stair package in case you did not purchase the Beam package. It uses the configuration settings for beams but will only detail channels and horizontal dimensions will always be below the beam. The end connections can be a single clip or two holes. Internal connections are limited to two holes per connection but there is no limit to the number of connections. If any blocks are required, the size of the block is entered manually. Where the blocks are dimensioned from depends on how you have the beam variable for blocks configured. It is assumed that the support at each end will be the back of a channel so the minus dimensions will be 0 for clip connections.

```
              Main mark: 1B1
Are marks RIGHT and LEFT (Yes/No) <N>: Enter
              Quantity: 1
(FT.IISS) Distance from back to back of landing channels: 7
Channel is toed (1)NS, (2)FS <2>: Enter
(FT.IISS) Top of channel to hole ......... #3: .0208
Sub assembly mark for clip: aa
Clip is (1)NS, (2)FS <1>: 2
(FT.IISS) Gage of clip: .0112
Add clip to shop bill (Yes/No) <Y>: Enter
(FT.IISS) Back of channel to hole ........ #1: .0204
(FT.IISS) End of channel to hole ......... #2 <1 1/4">: .0108
(FT.IISS) Top of channel to hole ......... #3: .0208
Do you want to enter a block size (Yes/No) <N>: Enter
Pick point, top of header beam left end:
Pick point to the right, top of header beam right end:
Is there a stringer connection (Yes/No) <Y>: Y
(FT.IISS) Back of support to back of stringer ... #1: 3
(FT.IISS) Back of stringer to first hole ........ #2: .0112
(FT.IISS) Top of header to first hole ........... #3: .0208
(FT.IISS) Hole to hole (enter 0 to omit) ........ #5: 0
```

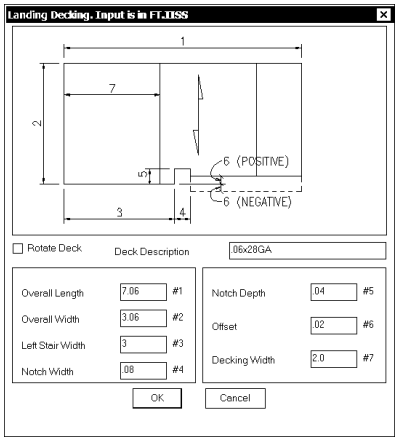
```
Pick loaction to draw holes for stringer connection:
Do you have another stringer connection (Yes/No): Y
(FT.IISS) Last stringer to next stringer ........ #4   <3'-0">: 1
(FT.IISS) Back of stringer to first hole ........ #2 <1 3/4">: -.0112
(FT.IISS) Top of header to first hole ........... #3 <2 1/2">: Enter
(FT.IISS) Hole to hole (enter 0 to omit) ........ #5: 0
Pick loaction to draw holes for stringer connection:
Do you have another stringer connection (Yes/No): N
Shop bill information on mark 1B1
Place shop bill text on LINE <4>: Enter
(FT.IISS) Thickness of connection material at LEFT end: .0009
(FT.IISS) Thickness of connection material at RIGHT end <9/16">: .0010
```

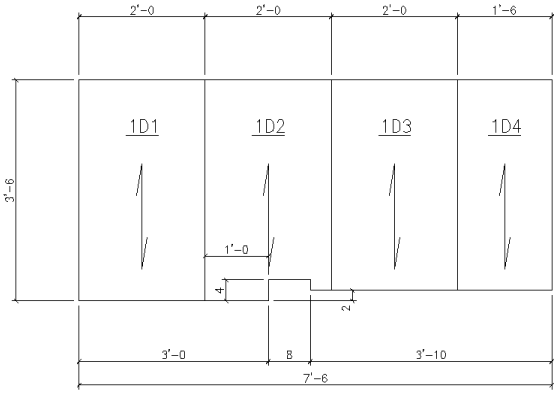
For internal connections the default is for the hole to be on the left side of the work line. To put the holes on the right side enter the distance as a negative.



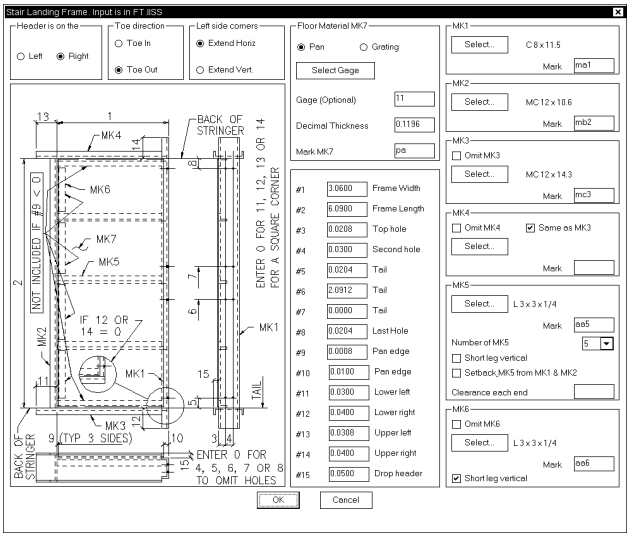
Landing Decking

This program is for detailing the decking that covers a stair landing. The deck length can run either direction. If the center notch is not required, enter 0 for dimensions 4 and 5. The layout is drawn to scale. Separate marks are given for each piece.





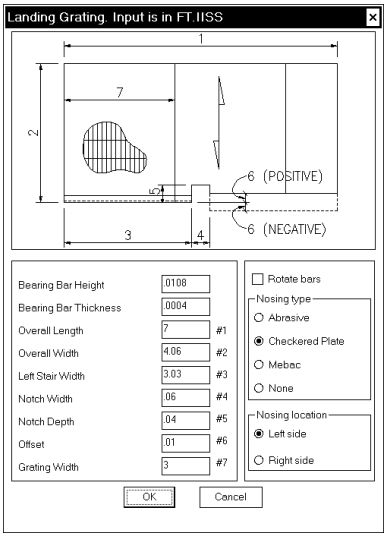
Landing Frame

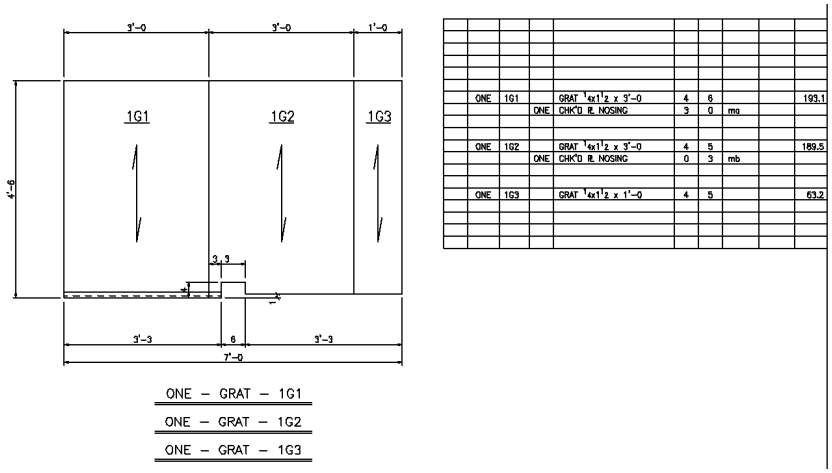


This program is for detailing shop welded frames out of channels with a pan shop attached or with grating field applied. Channels can be toed in or out. One channel can run past the corner, either vertically or horizontally or can stop even with the other. If channels are toed out and #15 is 0 then #9 may be negative to extend the pan over the flanges. You can omit MK6 for this condition. MK6 can run the full length as one piece or separate pieces between MK5. The thickness description for the Pan material that goes on top of the frame can be by Gage. Leaving the gage box blank will use the actual thickness in the description. If channels are toed out, you can omit MK3 and/or MK4 from the frame for cases where that channel is part of the stair stringer. All inputs are stored and recalled the next time the program is run.

Landing Grating

This program is for detailing the grating that covers a stair landing. Bearing bars can run either direction. If the center notch is not required, enter 0 for dimensions 4 and 5. The layout is drawn to scale. Separate marks are given for each piece.

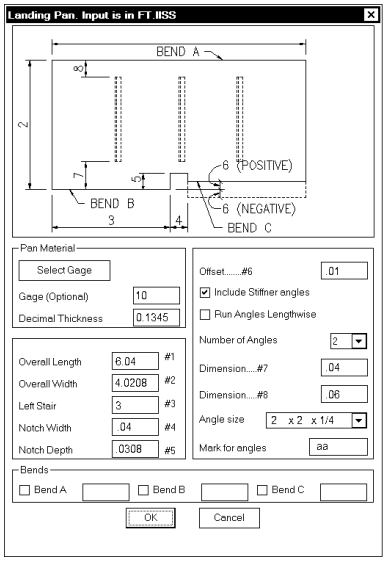


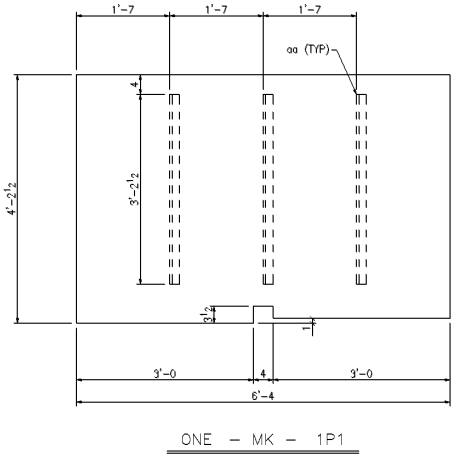


Landing Pan

This program is for detailing the pan or plate that covers a stair landing. Stiffener angles are optional and can run vertical or horizontal. If the center notch is not required, enter 0 for dimensions 4 and 5. Stiffeners will be equally spaced. For the Gage or Thickness, enter one or

the other but not both. This allows you to use a Gage material or a heavier plate. The layout is drawn to scale.





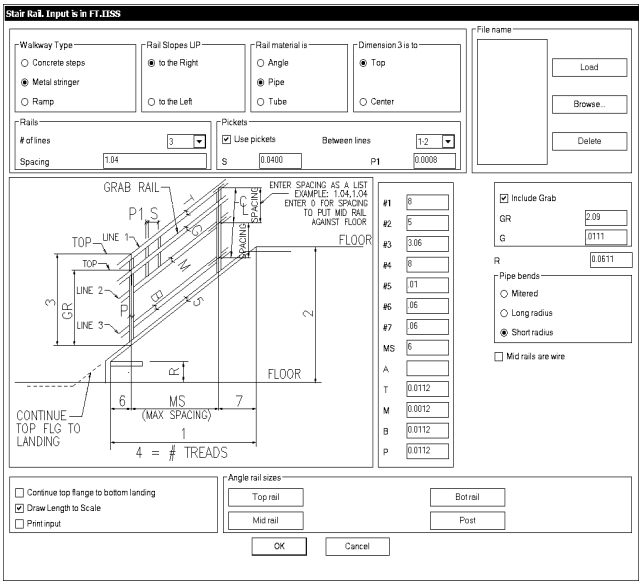
Chapter 13 - Rails

All programs in this chapter are under the AutoSD > Rails pulldown menu, Stairs / Rails Toolbar and AutoSD II > Rails ribbon menu.

RAMP or STAIR RAIL

With the Ramp / Stair Rail program you can draw single or multiple line ramp or stair rails with or without pickets, with or without grab rail using angle pipe or tube for the rail. For metal stringers the post can be bolted or welded. For ramps or concrete steps the post can be embedded or welded to an embed plate in the step or bolted using a base plate for ramp rails. Enter 0 for dimension "A" for the welded or bolted option. These rails are drawn to scale, to the actual bevel and sloping in either direction. The program is supported by icon menus and dialog boxes that guide you through the input.

`Command:` Pick **[**AutoSD or AutoSD II] > Rails > Ramp or Stair Rail



A dialog box is used to get your input. Select the options you want and enter the dimensions that are required. Some dimensions can be entered automatically from stair stringer input that has been saved. By using the list under "File name", select a file and pick **Load**. This list shows all of the stringer inputs that have been saved for this job. List from other jobs can be selected with the **Browse** button. The **Delete** button will delete a file from the current list.

The number of rail lines must be greater than 1 in order to enable the picket option. Pickets can be between any two pair of rails.

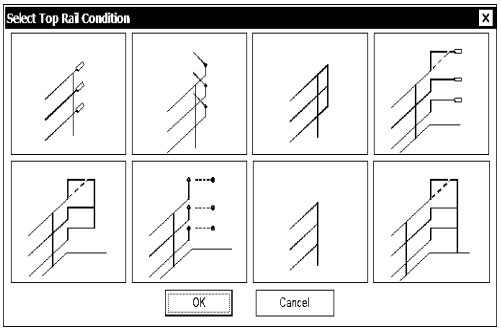
Spacing is the vertical distance from center line of rail to center line of rail. It can be entered as a single distance or as a list of distances separated by commas. If the number of distances is less than the number of spaces required for the # of lines, the last distance will be used for the remaining spaces.

#6 and #7 can be negative in order to put top or bottom post on the floor instead of on the stair.

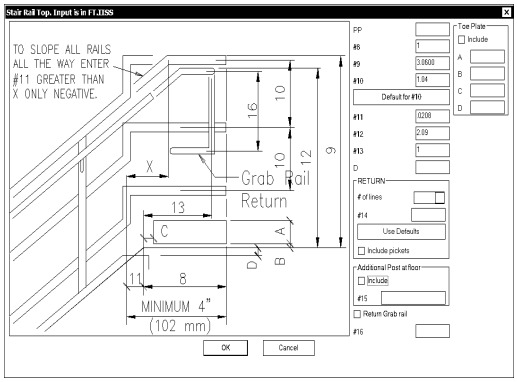
T, M, B, P and G are for the actual outside dimensions, not nominal.

Putting a check in the box for "Mid rails are wire" will add a vertical member where the rail bends.

Select OK when finished.

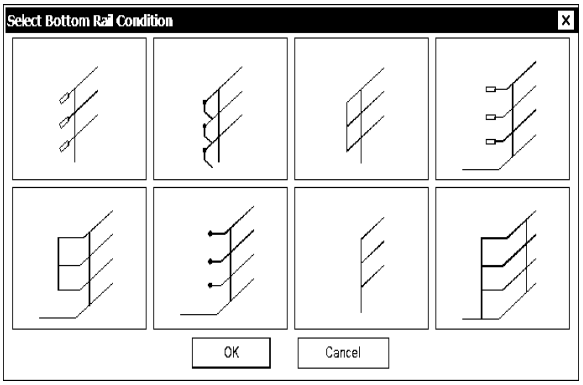


An icon menu appears
Pick the top condition

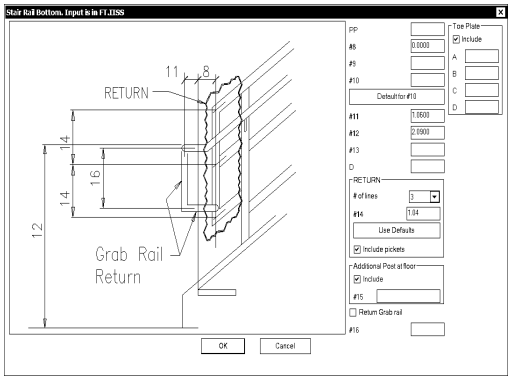


A dialog box of the top condition is shown for input

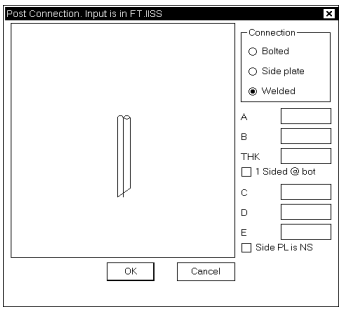
Type 6, (lower right corner), in the icon menus above is where the stair rail turns at the landing. Input for #10 and #14 follow the same rules as Spacing in the first dialog box. If you have a multiple line rail and want a single line at the end, beyond the post, you need to select an end that has a return.



Another icon menu appears
Pick the bottom condition



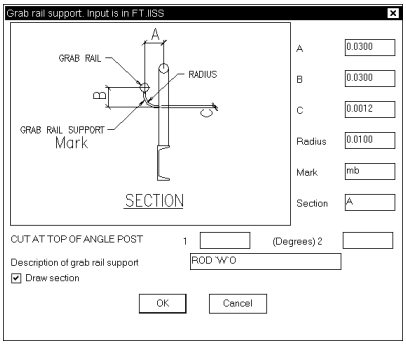
A dialog box of the bottom condition is shown for input. If #8 is the same as #6 then the rail would end at the post.



Pick "Bolted" if posts are bolted to the top of the stringer and enter dimensions for A, B and THK. Select “1 Sided @ bot” if the bottom post needs a plate with both bolts above the post and enter dimension for C, D and E.
Pick "Side plate" if posts have a vertical plate at the bottom of the post that are welded to the outside of the stringer and enter dimensions for A, B, C and THK and select if the plate is on the near side, (NS), of the post or not.
Pick "Welded", as shown, if posts are welded to the top of the stringer.

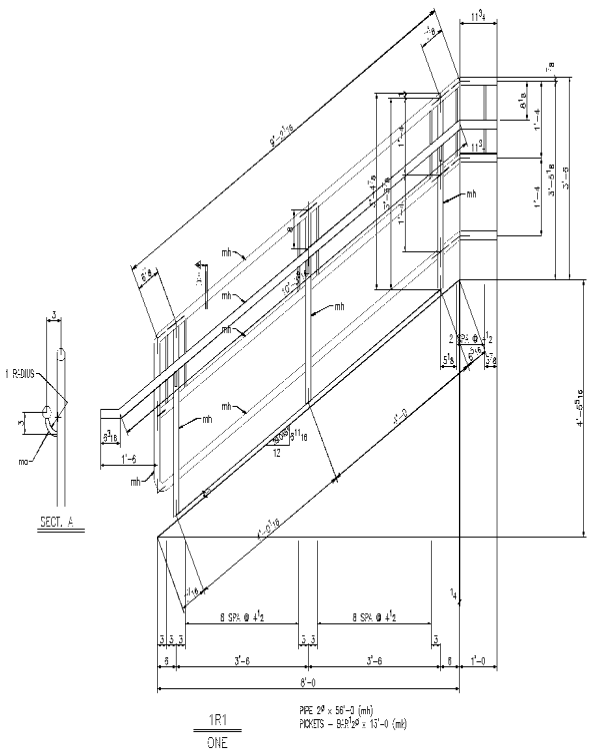
```
Grab rail is Near side or Far side (N/F): N
Insertion point:              A box is displayed around the cross hairs outlining
```

```
                                the area required for the rail detail.
End of rail at top is (Open/Splice) <O>: Enter
Insertion point @ section mark:
            Main mark: 1HR1
Are marks RIGHT and LEFT (Y/N) <N>: Enter
             Quantity: 1
Shop bill information on 1HR1
Place shop bill text on LINE <4>: Enter
Description of 2" O.D. rail: 1`8`O STD PIPE
Piece mark for 2`O STD PIPE
Enter mark or use <mh>: Enter
Description of pickets: BAR `8x`8
Piece mark for pickets
Enter mark or use <mk>: Enter
Command:
```

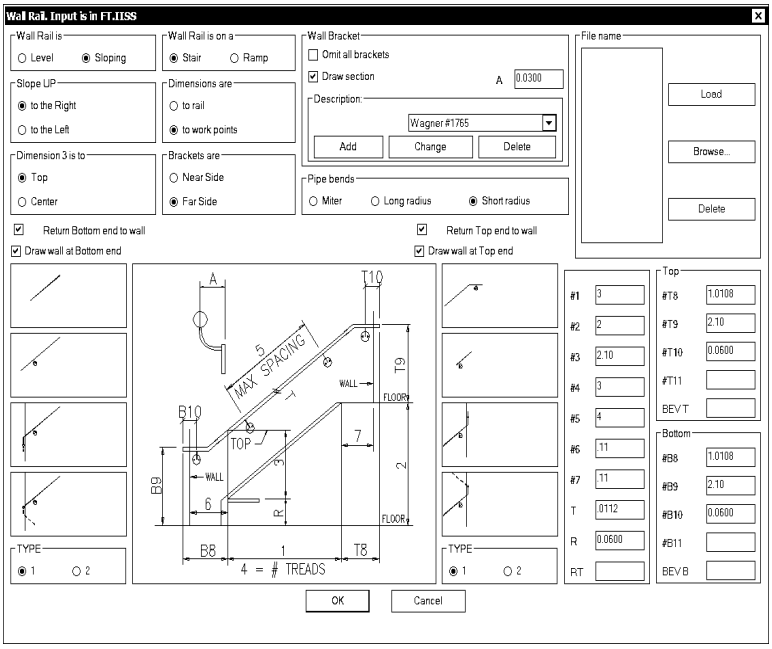


NOTE:

- If input #6 is negative the end post at the bottom will be on the landing.
- If input #7 is negative the end post at the top will be on the landing.
- If a grab rail is included and vertical input #12 is too high to work with horizontal input #11 or #13 to the end, the grab rail will not turn horizontally but will end where it crosses input #11 or #13.
- If you want the rail to end with a post use an end type with a return, (type 3 or 5), and the horizontal dimension from the work point to the return must be the same as the dimension from the work point to the post.
- If the rail ends in a splice, (type 1 or 4), the rail will extend a minimum of 4” past the post or bend.
- Number of rails must be greater than 1 to allow the option for pickets.
- The number of rails and vertical spacing for end types 3 and 5 can be different from the center section of rail.
- If you have a multiple line rail and want a single line at the end, beyond the post, you need to select an end that has a return, (type 3 or 5).
- Choose end type 4 or 5 if you want to include additional post at the floor level. The spacing of post is entered as a single distance or as a list of distances separated by commas. One dimension for each post space. If you want the program to calculate the post spacing using input “MS” as the max space then enter #6 or #7 as negative and the distance from the work point to the end post and do NOT check the box to include additional post.



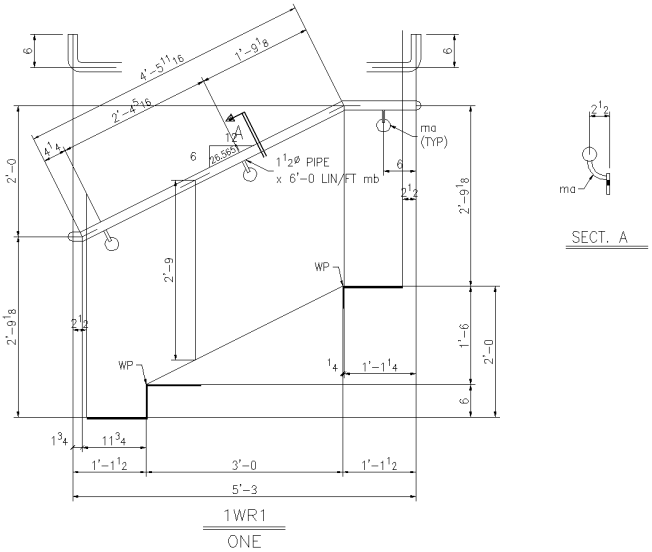
WALL RAIL



This program will draw wall rails. These rails are drawn to the actual bevel, sloping in either direction and the length may be drawn to scale.

A dialog box is used to get your input. Select the options you want and enter the dimensions that are required. Some dimensions can be entered automatically from stair stringer input that has been saved. By using the list under "File name", select a file and pick **Load**. This list shows all of the stringer inputs that have been saved for this job. List from other jobs can be selected with the **Browse** button. The **Delete** button will delete a file from the current list.

Insertion point: A box is displayed around the cross hairs outlining the area required for the rail detail.
Piece mark for wall rail support
Enter mark or use <ma>: Enter
Section mark for wall rail support: **A**
Pick location for section:
(FT.IISS) Enter dimension "A" for Top end <2 1/4">: **.06**
(FT.IISS) Enter dimension "A" for Bottom end <6">: **Enter**
Main mark: **1WR1**
Are marks RIGHT and LEFT (Y/N) <N>: **Enter**
Quantity: **1**
Shop bill information on 1WR1
Place shop bill text on LINE <4>: **Enter**
Piece mark for rail.
Enter mark or use <mb>: **Enter**
Shop bill information on mark mb
Description: **1`8`O PIPE**



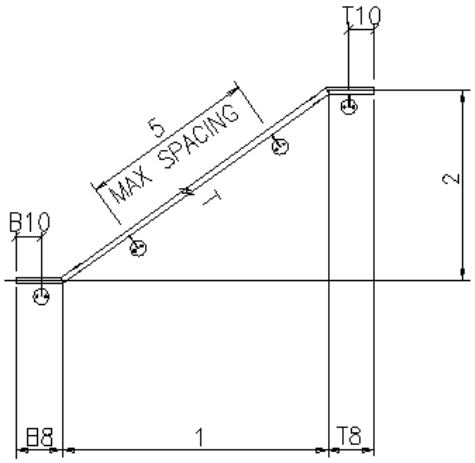
Wall Rail is: "LeveL" or "Sloping". Choosing "level" will draw a level length of wall rail that would be used above a floor. Choosing "sloping" will draw a wall rail for use on a stair or ramp.

Slope UP: "to the right" or "to the left". Select whether you want the rail drawn sloping up to the right or left.

Dimension 3 is to: "Top" or "Center". The height of the rail over the treads can be measured from the top or center line of the rail. The horizontal ends of the rail will always be measured to the top of the rail.

Wall Rail is on a: "Stair" or "Ramp": Choosing "ramp" reduces the number of options at the ends and the dimensions at the lower end do not include a riser for the first step of a stair.

Dimensions are: "to rail" or "to work points".



Choosing "to rail" allows you to enter the dimensions you want to use for the rail shown to the left. Choosing "to work point", as shown in the dialog box on the previous page, allows you to enter work lengths and have the program calculate the actual rail lengths.

Brackets are: "Near Side" or "Far Side". Select whether you want the section cut at the support brackets to show them on the near side or far side.

Wall Bracket:

Omit all brackets: Put a check in this box if you do not want any wall brackets**.**

Draw section: Put a check in this box if you want to draw the section of the support bracket.

Description: This is the description that will be used in the shop bill for the wall bracket. The first time you use the Wall Rail program the words "Type it" will be displayed in this box. To add descriptions pick **Add** and another box will pop up for you to enter a description. Pick OK in that box and your description will be added to the list and be displayed. If "Type it" is selected the program will ask for a description to use in the shop bill. If something else is selected then that will be used in the shop bill. Picking **Change** will allow you change an existing description. Picking **Delete** will allow you delete an existing description.

Pipe bends: Options are "Miter", "long radius" or "short radius". Select how you want the bends to be formed.

Return Bottom end to wall: Put a check in this box if you want a top view of the bottom end of the rail showing a return toward the wall.

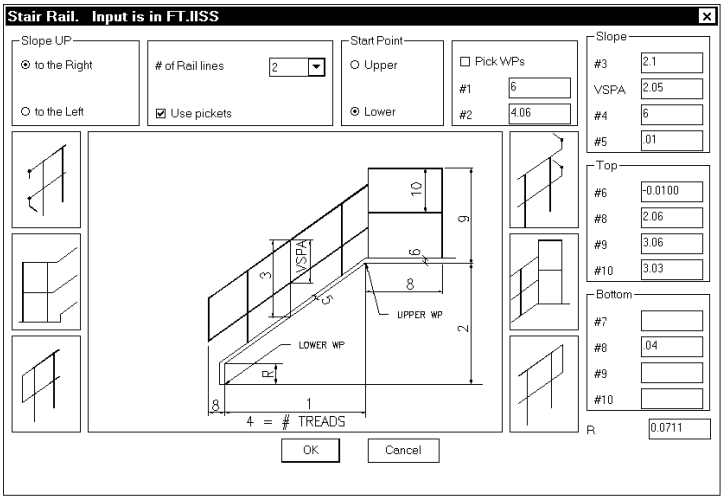
Draw Wall at Bottom end: Put a check in this box if you want to draw a reference line for a wall at the bottom end of the rail.

Return Top end to wall: Put a check in this box if you want a top view of the top end of the rail showing a return toward the wall.

Draw Wall at Top end: Put a check in this box if you want to draw a reference line for a wall at the top end of the rail.

Select the end condition for each end of the rail by picking the icons on either side of the large picture in the center. Enter the dimensions as required. If the bottom end turns and goes horizontally and if dimension B10 is greater than B8 the first bracket at the bottom will be on the slope at a distance from the bend of B10 - "horizontal length". The same is true at the top end.

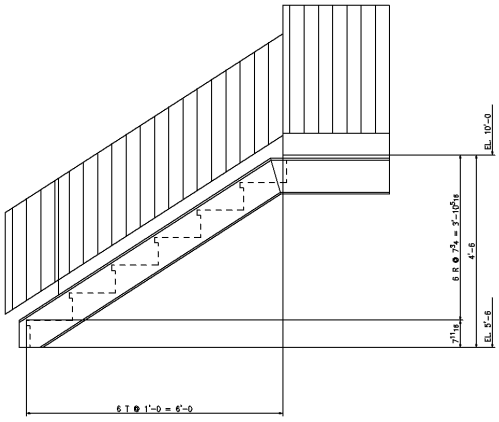
RAIL ELEVATION



After selecting the location for the vertical dimensions for the Stair Elevation the dialog box shown to the left pops up. It is for entering information to draw the elevation view of the handrail on the stair. This program can also be accessed from "AutoSD>Stairs/Rails>Erection> Rail Elevation". When popped up by the stringer program several options are pre-selected and disabled. Select the end conditions from the choices on each side of the main view and enter the dimensions required. Select Cancel if you do not want any handrail on the stair.

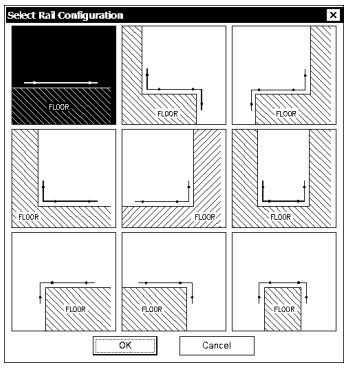
NOTE:

- This program is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command.
- The tread type shown in the lower right corner of the 3rd icon menu is for concrete steps.
- If you choose to pick the
- Lower work point, this point is on the finish floor on a vertical line through the noising point of the first tread.



LEVEL HANDRAIL

The Level Handrail program will draw single and multiple line level handrails, with posts bolted or welded, with or without pickets, with or without toe plate, using angles, pipes or tubes for the rail material. The height of the rail is drawn to scale and there is an option to draw the post spacing to scale. The program is supported by icon menus and dialog boxes that guide you through the input. You select where to start drawing the rail by selecting the location of the upper left corner of the detail. The handrail will be drawn from left to right with a top view above the portion shown horizontally in the icon menu. The handrail can be drawn looking from the outside, (toe plate far side), or from the inside, (toe plate near side), of the platform.

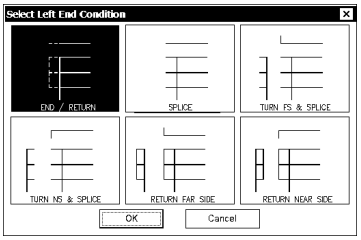



```
Command: Detail handrail viewed from the inside (Y/N) <Y>:
Enter     (Toe plate near side)
```

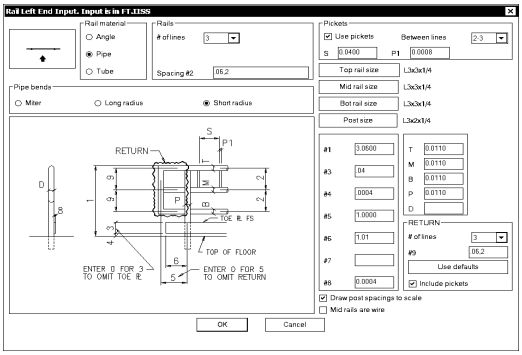
The icon menu to the left shows all of the possible handrail configurations. The one in the upper left corner will have one side. The others will have two or three sides as shown. Each side will have at least one post. In

addition to these configurations you will select one end condition from the icon menu below left for the extreme left and right ends. Some end conditions are for the rail to turn near side or far side. These turns will not have a post. If you need a handrail that looks like the one in the left middle without a post on the vertical side you would select the upper left icon and then select one of the end turns from the icon menu below.

The main mark will be placed under the horizontal portion shown in the icons above.



Choose the upper left icon shown in the icon menu to the left if you want to end the handrail in a post with or without a return. The icons that show a top view with a turn will allow you to turn the handrail at the ends without a post. These end types may be used with any of the handrail types shown in the first icon menu. If you have a multiple line rail and want a single line at the end, beyond the post, you need to select an end that has a return.

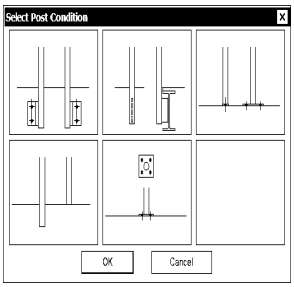


A dialog box is used to get your input. Select the options you want and enter the dimensions that are required. The image in the upper left indicates which portion of the rail is being detailed.
The number of rail lines must be greater than 1 in order to enable the picket option. Pickets can be between any two pair of rails but are not allowed on angle rails.

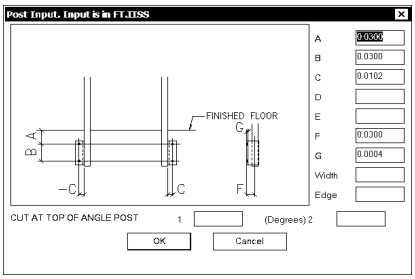
Spacing #2 is the vertical distance from center line of rail to center line of rail. It can be entered as a single distance or as a list of distances separated by commas. If the number of distances is less than the number of spaces required for the # of lines, the last distance will be used for the remaining spaces.

The # of lines and spacing #9 for ends with a return can be different from the main rail. T, B, P and D are for the actual outside dimension, not nominal.
Select OK when finished.

```
Insertion point at top left corner of rail:
```



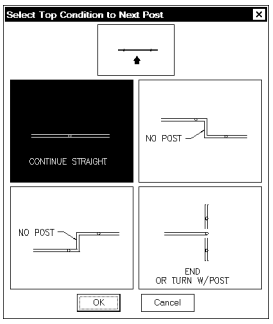
Select post connection



Enter input C as a negative to put the clip on the left side of the post.

```
Piece mark of toe plate
Enter mark or use <pa>: Enter
```

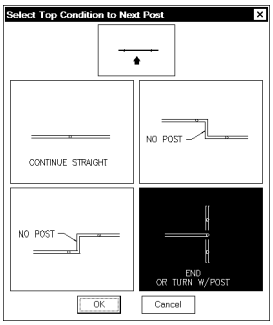
Section mark for post connection: **A**
Piece mark of Post clip
Enter angle mark or use <aa>: **Enter**

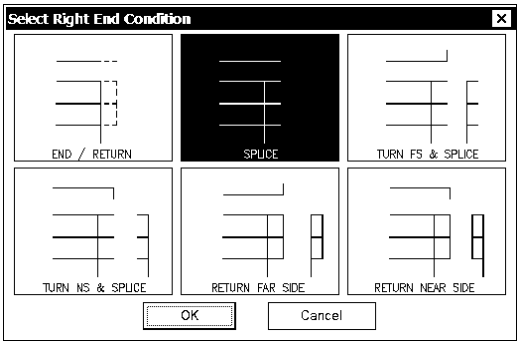


Select the upper left icon to continue going straight to the next post. Select the upper right or the lower left icon to zigzag to the next post. Select the lower right icon to end the handrail or turn a corner.
The image at the top center indicates the portion of the rail that is being detailed.

Select configuration of rail to next post

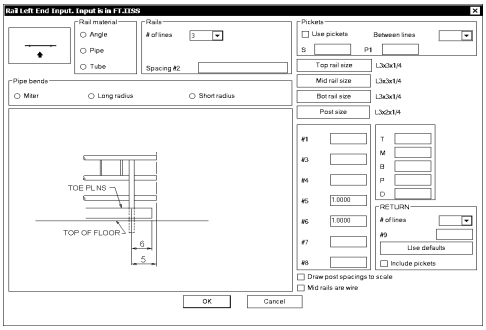
(FT.IISS) Distance to next post: 3
Is this post connection the same as the last (Yes/No) <Y>: **Enter**

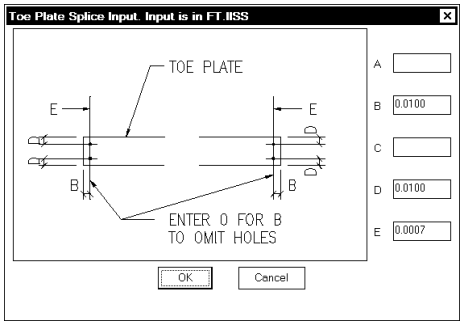




Select configuration of rail to next post

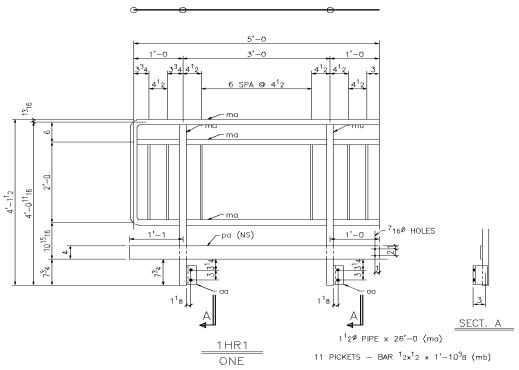
Select the right end condition





Enter values for toe plate splice connection. Enter 0 for B to omit holes in toe plate.

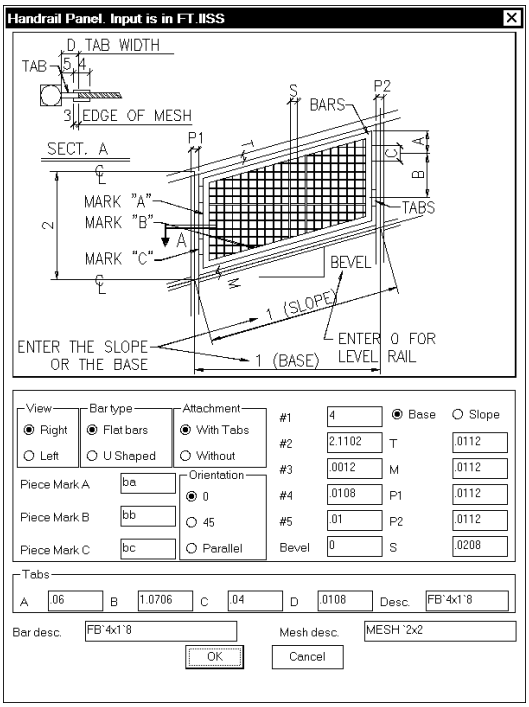
The dialog box for the right end is also used for the left end. Many of the inputs will be disabled. For this end type, only inputs for 5 and 6 are active. The image in the upper left corner indicates which portion of rail is being drawn.



End of rail is (Open/Splice) <O>:
The last mark was 1RR1
Main mark: **1HR1**
Are marks RIGHT and LEFT (Y/N) <N>: **Enter**

```
             Quantity: 1
Shop bill information on 1HR1
Place shop bill text on LINE <16>: Enter
Description of 1 3/4" rail: 1`8`O PIPE
Piece mark for 1`8`O PIPE
Enter mark or use <mg>: ma
```

Detail Panel



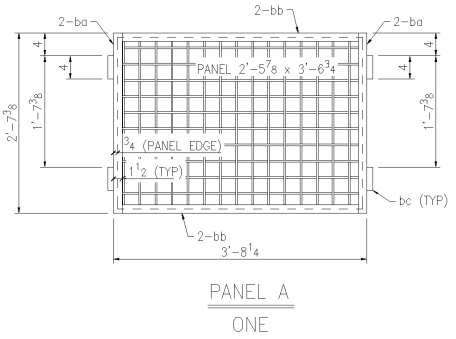
This program will detail and bill out a mesh panel insert for sloping or level handrail. If it is sloping it can be detailed sloped in either direction. Enter 0 for the bevel for a level rail panel.

View: Changes the direction the rail is sloped.
Bar type: Perimeter bars can be 2-Flat bars or 1-"U" bar.
Attachment of the panel to the rail can be wit or without tabs.
Orientation: The wire in the mesh can be horizontal, (0), 45 degrees or parallel to the top rail.

Dimension #1 can be entered as the Base or the Slope length. Select the one required by picking the option to the right of the #1 input box.

Enter a value for all of the fields that are enabled and pick OK.

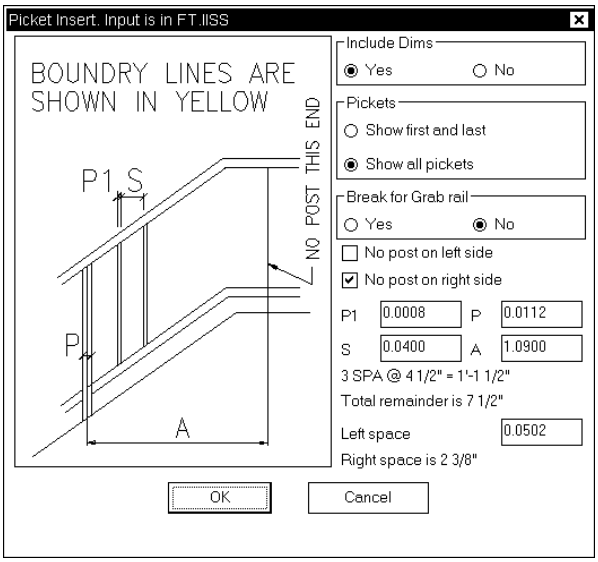
```
Command:
Insertion point at lower left corner:
The last mark was 1HR1
             Main mark: PANEL A
Are marks RIGHT and LEFT (Yes/No) <N>: Enter
              Quantity: 1
Shop bill information on mark PANEL A
Place shop bill text on LINE <9>: Enter
Shop bill information on mark ba
Shop bill information on mark bb
Shop bill information on mark bc
Command:
```



Insert Pickets

This program will draw pickets in an existing handrail detail for sloping or level handrail. If it is sloping it can be detailed sloped in either direction.

Include Dims: Check Yes to include horizontal dimensions.
Pickets: Select to show only the first and last or all of the pickets.
Break for Grab rail: Select Yes if you have a grab rail and the pickets are far side.
No post on left side: No post on right side: Check these options if the rail is open ended for a splice. Only one end can be open.
P1: picket width
P: post width
S: clear space between pickets
A: horizontal center to center limits of pickets. Enter a value and press the tab key. The number of spaces is calculated and a default is entered for the left space.
Left space: This is the space from the right side of the left post or picket to the left side of the first picket to the right. Enter a value and press the tab key and the right space is calculated and printed underneath.



When selecting the upper and lower borders select the face of the object the pickets are welded to.
When selecting the side borders select the center of the object on each side.

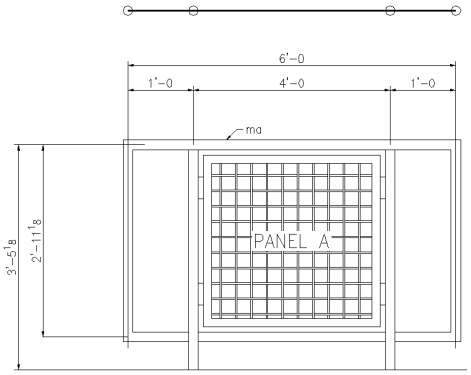
```
Command:
Select all boundary lines:                These may be selected in any order.
Pick dimension line location:             Pick a point.
Command:
```

Mesh Insert

This program draws a mesh panel within the lines selected. There is no limit to the number of lines for the border. Any of the boundary lines may be picked first but the remainder MUST be selected in a counterclockwise order.

```
Command:
Select all boundary lines picking a counterclockwise rotation.
Select objects: 1 found
Select objects: 1 found, 2 total
Select objects: 1 found, 3 total
Select objects: 1 found, 4 total
Select objects:
Panel Mark: A
(FT.IISS) Width of bars around perimeter: .0108
(FT.IISS) Size of openings in Mesh: .0208
Rotation of mesh (0/45/Select): 0
Attach with tabs (Yes/No): Y
```

(FT.IISS) Gap between post and bars: .01



1HR1
ONE

1 1/2Ø PIPE x 26'-0 (ma)

Chapter 14 - Ladders

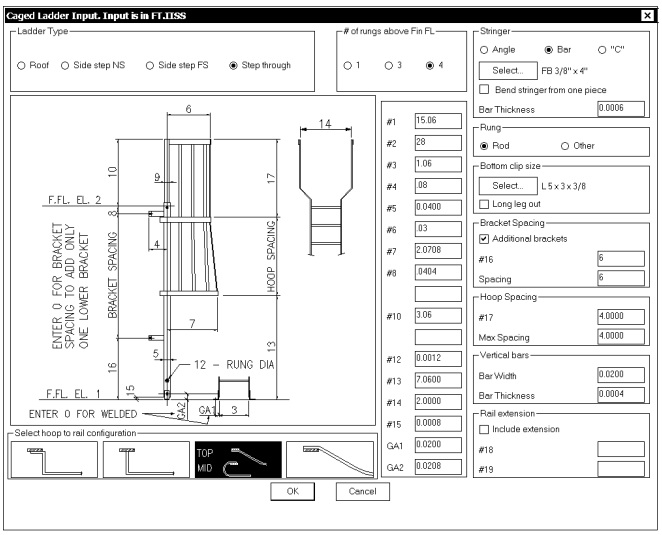
All programs in this chapter are under the AutoSD > Ladders pulldown menu, Ladders toolbar and AutoSD II > Ladders ribbon menu.

Ladders can be detailed with or without cages. Ladders without cages can be wall mounted or sit on the floor and also bolted to the wall. Some can go over a parapet and others can go in an elevator pit or up to a roof hatch.

Ladders with cages can be roof hatch, side step near side, side step far side or step through. The ladder itself is a block that is inserted and dimensions are added to the block based on your input. Hoops are drawn to scale and the shop bill is filled out for you.

LADDERS WITH CAGE

Pulldown: AutoSD > Ladders > Ladders W/Cage



Change ladder types by selecting Roof, Side step or Step through. "# of rungs above Fin FL" is for Side step type. Enter input #1 - #15 as required. If GA1 is 0, the hole in the OSL will be omitted. If GA2 is 0, the clip will be welded to the ladder.

Stringer: Select Angle, Bar or "C" for channel. If an angle or channel is used, pick the "Select" button to select the stringer size. If a bar is used, enter the thickness. If the ladder is a step through, and the stringer is a bar, put a check in the box "Bend stringer from one piece" if you do not want the stringer bar to be welded from two pieces.

If the stringer is an angle or channel, put a check in the box "Stringer at step through is a bar" if that is what you want, otherwise, the stringer at the step through will be the same material as the main stringer.

Rung: Select Rod or "Other". The "other" option allows you to enter a different description for the rung and will be detailed as a "C" shape.

Bottom clip size: Pick the "Select" button to select the clip size. Put a check in the box if the long leg is outstanding for unequal leg angle.

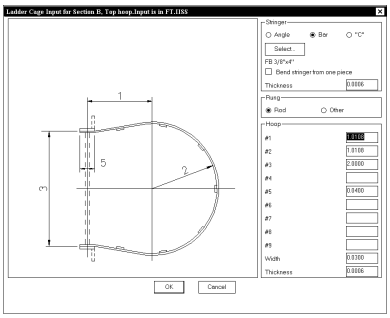
Bracket Spacing:

Additional Brackets: Put a check in the box if you need more stringer supports.

Spacing: There are four ways to enter the spacing. **1)** Enter 0 to have only one additional bracket a distance of input #16 from the bottom floor. **2)** Enter a single dimension greater than 0, (example: 6.06), to add equally spaced brackets between the bottom and top brackets where the spacing is less than or equal to your input. **3)** Enter multiple dimensions separated by a comma, (example: 4, 5, 4.06), to add brackets spaced at 4'-0, 5'-0 and the remaining space divided equally where the spacing is less than or equal to the last dimension of your input. **4)**

Enter multiple dimensions separated by a comma with the last dimension 0, (example: 4, 5, 4.06, 0), to add brackets spaced at 4'-0, 5'-0 and 4'-6 with no additional brackets except for the one at the top.

Hoop Spacing: #17 is for the space from the top hoop down to the second hoop. The remaining distance between #13 and #17 will be divided into equal spaces where the spacing is less than or equal to the Max Spacing.

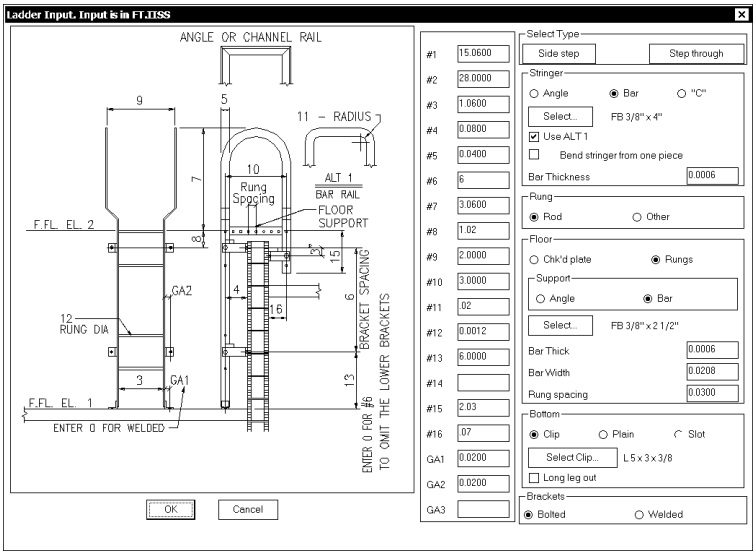


After picking OK and entering the main mark and quantity, a dialog box is used for input of the hoops. The stringer size defaults to that chosen in the ladder input dialog box. Enter input #1 - #9 as required and the hoop Width and Thickness and pick OK. A rectangular outline box is used to help in placing the detail.

The top hoop is drawn, billed out and this process is repeated for the mid hoop, Bottom hoop and sections.

LADDERS WITHOUT CAGE

Pulldown: AutoSD > Ladders > Ladders WO/Cage



Change ladder types by picking a Select Type button for Side step or Step through. Enter input #1 - #16 as required. If GA1 is 0, the hole in the OSL will be omitted. GA2 and GA3 cannot be 0.

Stringer: Select Angle, Bar or "C" for channel. If an angle or channel is used, pick the Select button to select the stringer size. If a bar is used, enter a thickness. #5 is used for the width. If the ladder is a step through, and the stringer is a bar, put a check in the box "Bend stringer from one piece" if you do not want the stringer bar to be welded from two pieces.

If the stringer is an angle or channel, put a check in the box "Stringer at step through is a bar" if that is what you want, otherwise, the stringer at the step through will be the same material as the main stringer.

Rung: Select Rod or "Other". The "other" option allows you to enter a different description for the rung and will be detailed as a "C" shape.

Floor: This option is available for the step through parapet ladder. The floor can be either checkered plate or rungs. The floor support can be a bar or angle. If an angle is used, pick the

Select button to select the size. If a bar is used, enter the Width and thickness. The Rung spacing is for the horizontal spacing of rungs used for the floor.

Bottom: Select Clip, Plain or Slot for the connection of the bottom to the floor. If an unequal leg clip is used, put a check in the box if the long leg is outstanding.

Brackets: May be bolted of welded for parapet ladders only.

#6 - Bracket Spacing: There are four ways to enter the spacing. **1)** Enter 0 to have only the one bracket at the top. **2)** Enter a single dimension greater than 0, (example: 6.06), to add equally spaced brackets between the bottom and top brackets where the spacing is less than or equal to your input. **3)** Enter multiple dimensions separated by a comma, (example: 4, 5, 4.06), to add brackets spaced at 4'-0, 5'-0 and the remaining space divided equally where the spacing is less than or equal to the last dimension of your input. **4)** Enter multiple dimensions separated by a comma with the last dimension 0, (example: 4, 5, 4.06, 0), to add brackets spaced at 4'-0, 5'-0 and 4'-6 with no additional brackets except for the one at the top

After picking OK and entering the main mark and quantity, the ladder is drawn and billed out.

TOP HOOP (SECT B)


Pulldown: AutoSD > Ladders > Top Hoop (Sect B) > Side Step

Pulldown: AutoSD > Ladders > Top Hoop (Sect B) > Step Through
Top hoops can be drawn separately if needed.

MID HOOP (SECT C)


Pulldown: AutoSD > Ladders > Mid Hoop (Sect C)
Mid hoops can be drawn separately if needed.

BOT HOOP (SECT D)


Pulldown: AutoSD > Ladders > Bot Hoop (Sect D)
Bottom hoops can be drawn separately if needed.

SECTIONS


Pulldown: AutoSD > Ladders > Sections
Various sections can be added if needed.

SECURITY GUARD

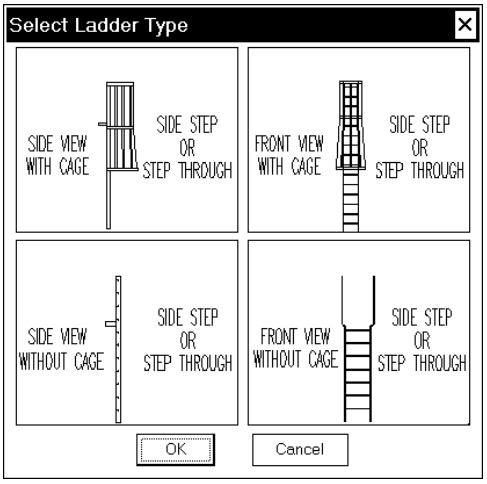

Pulldown: AutoSD > Ladders > Security Guard

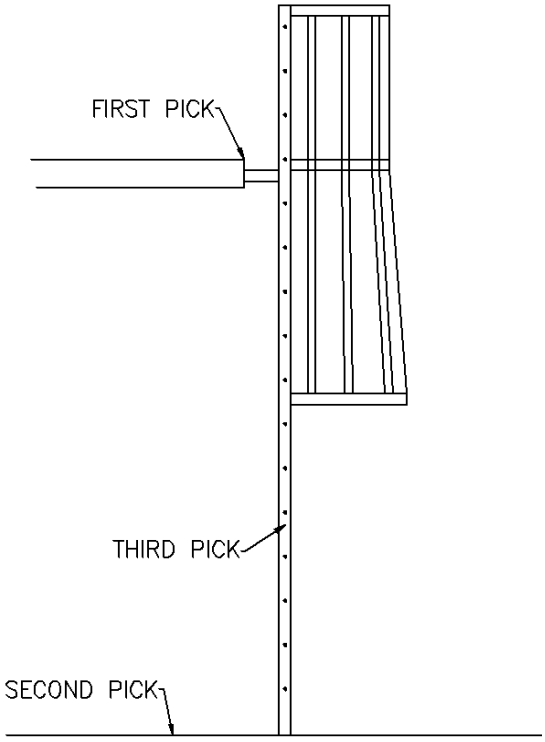
Add a Security guard detail to an existing ladder detail. The guard may be made from sheet metal or an angle frame with wire mesh. The hinges and latch may be on either side.

Insert reference: If you check this option a rectangle representing the outline of the security guard will be inserted using the lower left corner as the insertion point. You will be asked if the ladder detail is a front view or a side view. Pick a point on the ladder stringer near the bottom to insert the reference outline.

ERECTION ELEVATION

This command is for drawing the elevation view of a ladder on an erection drawing. There is a caged and non-caged ladder. Both can be drawn in front view or side view. All ladders can be drawn as a side step or step through. The programs require three points to locate the ladder. The first pick is the top of the upper floor. If you are drawing the side view, this pick should be the outside corner of the landing. The second pick is the top of the lower floor. The third pick is the center line of the ladder rails.
The scale is affected by the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command

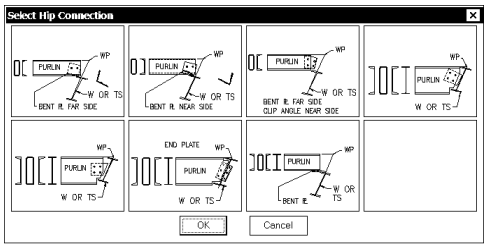


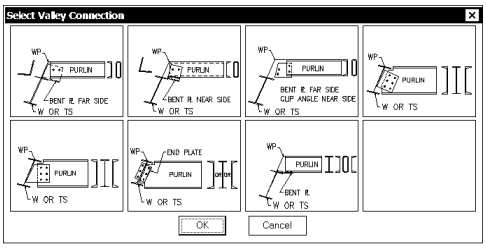


Chapter 15 – Hip & Valley

All programs in this chapter are under the AutoSD > Hip and Valley pulldown menu and AutoSD II > Hip&Valley ribbon menu.

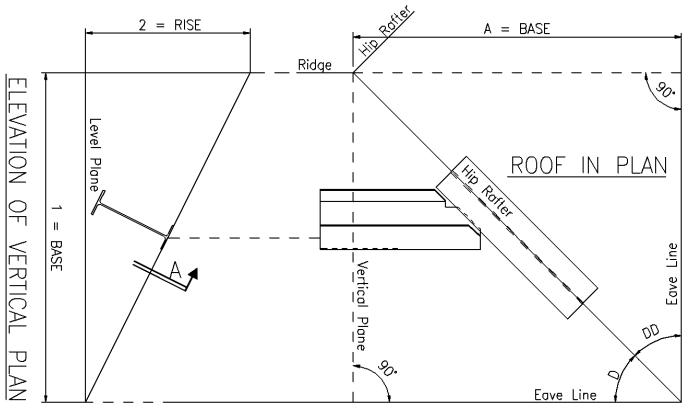
The HIP and VALLEY programs will design connections where the purlin is horizontal, (parallel to the ground), and the web is tilted, (90^O to the roof plane). They will draw a complete layout of the purlin and rafter with section views showing detail dimensions of the connection. This layout will take up the entire drawing; therefore, your drawing should be blank before you start. Start a new drawing and run setup first, setting the drawing size to 24" x 36" and the scale to 1"=1'-0. Purlins can be sitting on top of the rafter or framing into the side. All views are drawn to scale for easy placement of the purlin connections. Easy selection of member sizes is made from dialog boxes.





There are seven different hip and valley connections to choose from. The first three and the last one shown in the icon menus above have the purlin sitting on top of the rafter. The shapes for purlins and rafters are noted in each icon menu.

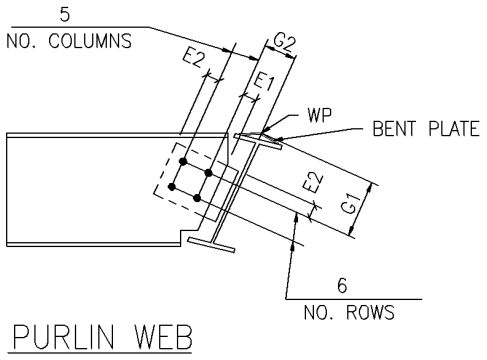
The example below is the hip connection shown in the lower left corner of the icon menu above.



A slide is displayed for user reference.

```
(FT.IISS) Enter Base control dim #1: 22
(FT.IISS) Enter Rise control dim #2: 11
(Decimal) Enter Horiz control angle D: 45
```

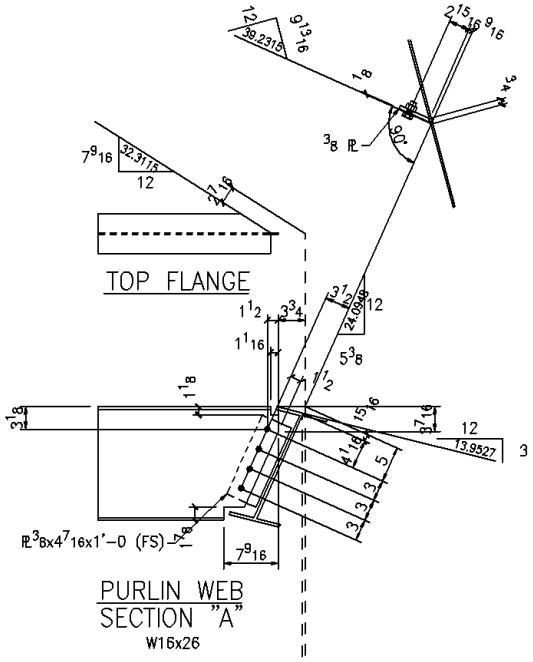
```
(Decimal) Enter Horiz control angle DD: <45.0000> Enter
```



A slide is displayed for user reference.

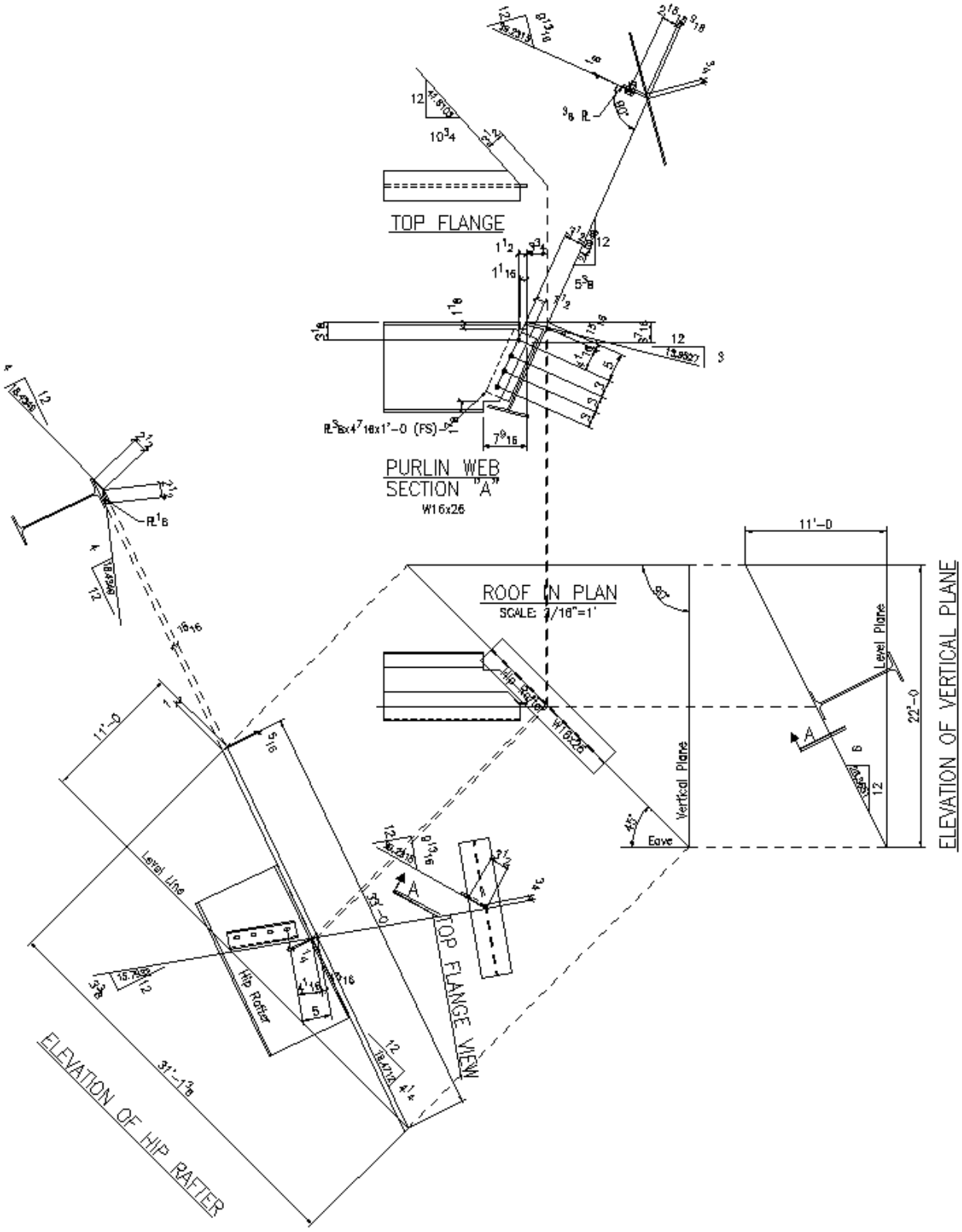
```
(FT.IISS) Thickness of connection plate: .0006
Enter Number of columns of holes, #5: 1
Enter Number of rows, (Max will fit is 5), #6: 4
(FT.IISS) Enter edge distance at end of purlin E1: .0108
(FT.IISS) Enter edge distance on plate E2: .0108
(FT.IISS) Enter WP to first hole, G1, |Min = 4 7/8"|: .05
(FT.IISS) Enter WP to first hole, G2, |Min = 3 1/2"|: .0308
(FT.IISS) Thickness of bent plate on top flange: .0002
```

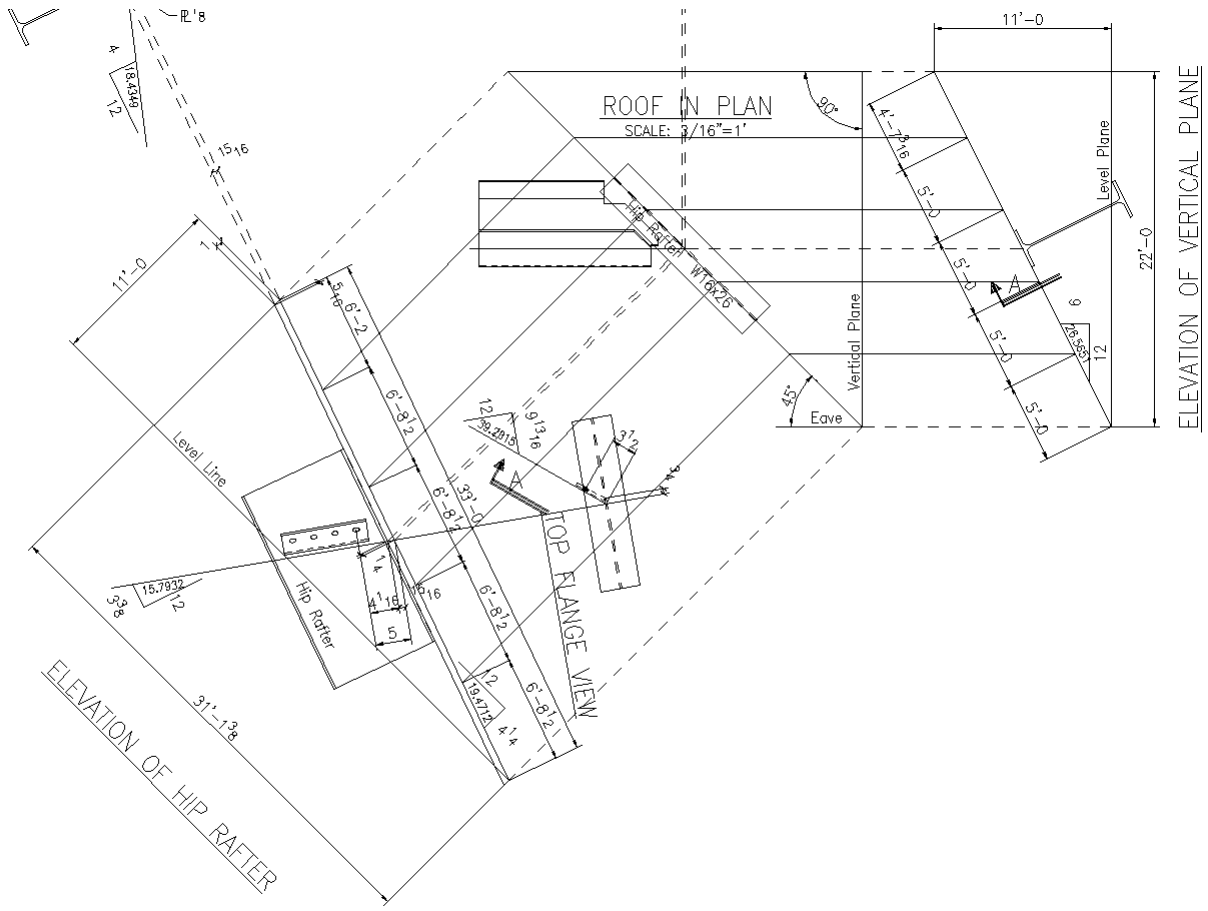
Some of the prompt lines in different connections will show things like |Min = 4 11/16"|, (Min will fit is 3 1/2"), (Min is 3 5/16") or |Use 5 1/16"|. These are prompts giving recommended minimum or maximum dimensions based on member sizes selected and dimensions enter before the prompt. They are meant to be used as guides for entering information.



A section is cut showing the connection detail of the purlin giving enough dimensions to be able to detail the purlin as well as the connection on the rafter.

A complete layout is generated as shown below. Be sure to zoom in on Section “A” and examine this section closely for any interference.

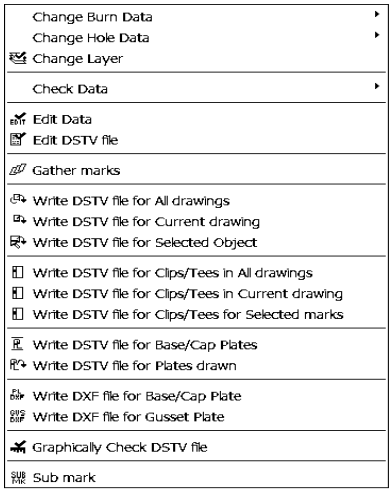




In the plan view there is a scale given. In this case it is 3/16 = 1’-0. The scale will vary depending on the dimensions given for #1 and #2 in order to keep the drawing a certain size. You can use the [AutoSD > Modify or AutoSD I > Change] > Scale of Drawing command to set the drawing to this scale. After doing this you can draw a line 90 degrees to the roof plane line at one end in the elevation of vertical plane view and then use the offset command to lay out the purlin spacing in this view and project those spaces to the plan view and elevation of hip rafter view. The example above shows a spacing of 5’-0 starting at the eave. This will allow you to develop the locations of the connections on the rafter.

Chapter 16 - CNC Data

Pulldown: AutoSD > CNC
Toolbar: CNC
Ribbon: AutoSD II > CNC



CNC data will be attached as extended data, (xdata), to holes when you detail a beam, girt, column, brace or stair stringer and burn information will be attached to blocks and skewed ends when you detail a beam, girt or stair stringer if you have configured your customer to save CNC data.

Xdata for one member is separated from another member by using different layers. A layer will be created with the same name as the main mark or sub mark and prefixed with "CNC_" when a member is detailed by the program. All entities with CNC data for this detail will be on this layer. If you copy an existing detail to make a new one you MUST create a new layer with the name of the layer the same as the new mark, prefixed with "CNC_", set it current and change all entities of the new detail to that layer. You can do all of this with the command "Change CNC layer" under "AutoSD > CNC". If the layer name does not exist it will be created, if it does exist it will be set current. All entities selected will be changed to the new layer

Stairs offer a different problem with separating the xdata since a stair main mark can have two stringers and one stringer can be made from two or more pieces. To handle these conditions a layer will be created for each sub piece. The layer name will be the sub piece mark prefixed with "CNC_". For example, if the sub piece mark is **ma**$^{\mathbf{R}}$ on sheet 43 the layer name will be CNC_ma and the CNC file name will be 43maR and 43maL. You can also use this layer naming convention for built up frames made out of flange shapes. Since frames are not generated by a program you will need to create the layers and change the objects layer after detailing is complete or set the correct layer current before detailing each member..

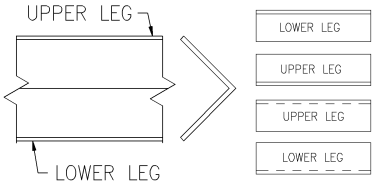
When xdata is extracted from the drawing, a file with the same name as the main mark will be created to hold the CNC data for that piece. These files will be saved in the same folder the drawing is in. If the main mark or sub mark for a detail with CNC data starts with a letter instead of a number, the CNC file name will start with the drawing number. The CNC file name for detail A on sheet 3 will be 3A. Marks can be rights and lefts. If you use "Opposite hand" for a mark, the quantity must include “`G” or “~G” for a left.

Tail dimension text will be associated with the holes that the tail dimension is for. If you change the tail dimension text using the command "Replace Dim" under "Modify/Change" the tail dimension stored in the xdata will also be changed. If you have two sets of holes and tail dimension text to only the first hole the xdata for the second set of holes will NOT be changed when you change the tail dimension text. You must change the tail dimension stored in the

xdata of the other holes with the command "Change Tail xdata" or use the “Edit Data” command. All tail dimensions must go to end of material.

MAIN MARK

The Main mark will be a block with attributes for the mark, quantity, member size and length.



The member size and length attributes will be invisible. If you edit the member size or length on the description line of the detail you will also need to edit this information in the main mark tag by double clicking on the mark or with the command attedit or battman.

Angles: Angles may be detailed in any orientation. The picture to the left indicates which leg is the “Upper” and which is the “Lower” leg. If the angle legs are not equal the description in the piece mark attribute must be entered as “L” Upper leg “x” Lower leg “x” thickness.

HOLES

The data information saved with each hole will be the face the hole is in, the gage of the face, which side from the face center line the hole is in, the hole size, the tail dimension, whether the member is a "Beam", "Angle", "Column, or Brace", which end the tail is from and which edge the gage is from. Xdata for "Column and Brace" is treated differently from other members because holes in the web are measured from the centerline and holes in the web of other members are measured from the flange or heel.

Since the xdata is stored in the hole entity, if you erase a hole you erase the xdata for that hole. Holes with xdata attached will be drawn using a solid polyline. Holes without xdata attached will be drawn using several lines close together. This way you can tell if xdata has been saved with a hole just by looking at the hole.

ASSOCIATE HOLES with TAIL

This program is used after you insert a hole for an existing tail dimension to tie the hole tail dimension xdata to the tail dimension text.

When a tail dimension is placed to a hole, the text is associated with the xdata of the hole so that when the dimension text is changed the xdata for the hole is also changed. If there is more than one hole associated with a tail dimension text, the xdata for all of the holes will be changed when the tail dimension text is changed.

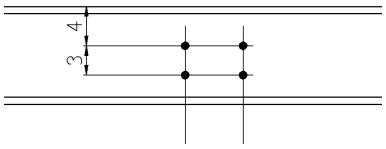
The following commands are shortcuts for changing one value of hole data but multiple holes can be changed at one time. For a complete look at the hole data use the Edit Data command.

CHANGE FACE

This program changes the xdata "face" value of all holes selected. The face values for beams are top flange, web and bottom flange. The face values for columns are left flange, web

NS&FS, web NS, web FS and right flange. The face values for angles are upper leg and lower leg.

GA CHANGE GAGE



This program changes the xdata "gage" value of all holes selected.
Beam web: gage = distance down from the top flange or up from bottom flange.
Column or W brace web: gage = distance from hole to hole on each side of the center line of the web.
Beam or Column flange: gage = the given gage as detailed.
Angle: gage = the given gage as detailed.

The gages in the picture shown above are 4" and 7".

OPT CHANGE OPTION

This program changes the xdata "option" value of holes selected. The available values are for:
Tail from left end, gage from top flange
Tail from right end, gage from top flange
Tail from left end, gage from bottom flange
Tail from right end, gage from bottom flange

CHANGE SIZE

This program changes the xdata "hole size" value of all holes selected.

CHANGE SIDE

This program changes the xdata "side" value of all holes selected. The side value for a beam and column flange is near side, (NS), far side, (FS) or both sides, (BS).
The side value for a column web are left side of center line, right side of center line and both sides of center line. The "side" value is not used if the shape is an angle or channel.

CHANGE SLOT ANGLE

This program changes the xdata angle value of all slots selected.

CHANGE SLOT LENGTH

This program changes the xdata length value of all slots selected.

CHANGE SLOT WIDTH

This program changes the xdata width value of all slots selected.

TAIL

CHANGE TAIL

This program changes the xdata "tail" value of all holes selected.

CHANGE HOLE TO SLOT

This program changes the xdata of all holes selected to slots and redraws the hole as a slot. You must enter the slot length, width and angle.

CHANGE SLOT TO HOLE

This program changes the xdata of all slots selected to holes and redraws the slot as a hole.

NS
FS
BS

NOTE HOLES NS, FS or BS

These are for labeling holes using a leader line. You start with the location for the arrow. The "NEAR" snap is automatically activated for this selection. Pick the next point for the leader line and continue picking points. Press Enter after selecting the last point and the note will be added to the end of the line.

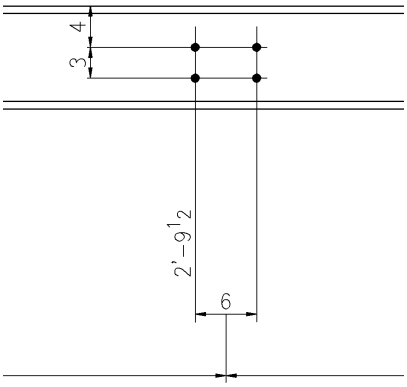
If the first leader segment crosses any lines you will be asked if you want to select lines to omit arrowheads. If you answer No arrows will be put at each intersection between the first leader segment and each line it crosses.

If you are saving CNC data you are prompted to "Select objects". You should select all the holes that this note references. The CNC data will be automatically updated. This note is NOT associated with the holes. If you change the note the hole data will NOT be updated.

REP

REPLACE TAIL DIMENSION

Pulldown: AutoSD > Modify
Toolbar: Change
Ribbon: AutoSD I > Change



This program is used to change the tail dimension text. If the text is associated with some holes the tail dimensions stored as xdata for the holes will also be changed. The tail dimension text will only be associated with holes that the dimension is for. In the picture shown to the left the 2'-9 1/2 dimension text is associated only with the two holes on the line the text is on. The other two holes have no association with any dimension text. If you change the dimension text you will need to use the command "Change Tail xdata" to change the xdata dimension stored with the other two holes at 3'-3 1/2. The same program is also located under Modify/Change as Replace Dim.

BURNS

The data information saved with each burn depends on the type of burn and can be viewed and edited.

Xdata for burns is stored with a line or a text entity. If you erase that entity you erase the xdata for that burn. Most burn xdata is stored with the horizontal length text for that burn. Skewed flange, skewed web burns, contours and cutouts are stored with the line for the edge of the burn.

The following commands are shortcuts for changing one value of burn data but multiple burns can be changed at one time. For a complete look at the burn data use the Edit Data command.

CHANGE ANGLE

If this is for beveling the flange for a groove weld the choices are (+-)30 and (+-)45. If this is for a skewed cut across the flange or web, it is the angle normal to that surface. For skewed cuts, if X or Y is 0 the cut will be full width using the angle. If X and Y are not zero the skewed cut will be based on X and Y and the angle will be used only to determine if it is positive or negative. In this case the angle could be 1 or -1. For Blocks it is the angle of the beam slope. The angle can be positive or negative.

CHANGE BEAM END

This changes which end of the beam the burn is on. Available choices are (1) LEFT and (2) RIGHT.

CHANGE DEPTH “Y”

This will change the vertical depth of the burn.

CHANGE LENGTH “X”

This will change the horizontal length of the burn. This will not change the dimension text. It will only change the xdata. Use “Replace Tail Dimension” under AutoSD > CNC > Change Hole Data or “Replace Dimension” under Modify/Change to change both Length text and Length xdata..

CHANGE NOTCH “D”

This will change the vertical distance from the top flange of the beam to the top edge of the burn for a notch in the web.

CHANGE SETBACK

This will change the distance from the clip to the end of the web.

CHANGE SIDE

This is for flange thinning burns and will change which side of the web the flange will be cut. Available choices are (1) NS, (2) FS and (3) Both sides.

CHANGE SURFACE

This will change the surface the burn is in. Available choices are (1) Top flange or Lower leg, (2) Bottom Flange or Upper leg and (3) Web.

CHANGE TYPE

This will change the type of burn. Available choices are "Bevel Prep", "Beveled Flg", "Block", "Contour", "Cutout", "End Notch", "Flg Notch", "Flg Thinning", "Rat Hole", "Skewed Cut" and "Slot".

CHANGE LAYER

If you change the mark of an existing detail to make a new one you MUST create a new layer with the name of the layer the same as the new mark, prefixed with "CNC_", set it current and change all entities of the new detail to that layer. You can do all of this with this command. If the layer name does not exist it will be created, if it does exist it will be set current. Entities on layers ASD_SECT, ASD_MARK, HIDDEN and HIDEN will not change layers. These entities include main marks, sections marks and hidden lines and should remain on these layers.

CHECK DATA

All Drawings: This will extract the CNC xdata from all drawings in a selected folder.
Current Drawing: This will extract the CNC xdata from the current drawing.
Single object: This will extract the CNC xdata from the details selected. Select the main mark under the detail for beams, columns and single member bracing. Select the sub mark for stair stringers and double angle bracing.

The CNC data information will be written to a text file in a format that is easy to read. The file name will be CNC_CHECK.txt and will be put in the folder the drawing is in.

Drawing : 1
Mark : 1B1
Member : W16X26
Length : 21'-10 1/2"
Quantity: 1
Grade: A36
Total number of holes: 20

----- Holes -----

Face	Tail Dim	Gage	Diameter	Side	Tail from,	Gage from
Web	1 1/2"	3"	13/16"		Left end	Top flg
Web	1 1/2"	6"	13/16"		Left end	Top flg
Web	1 1/2"	9"	13/16"		Left end	Top flg
Web	4 1/2"	3"	13/16"		Left end	Top flg
Web	4 1/2"	6"	13/16"		Left end	Top flg
Web	4 1/2"	9"	13/16"		Left end	Top flg

Web	5'-11 1/2"	3"	13/16"	Left end	Top flg
Web	5'-11 1/2"	6"	13/16"	Left end	Top flg
Web	6'-5"	3"	13/16"	Left end	Top flg
Web	6'-5"	6"	13/16"	Left end	Top flg
Web	9'-5 1/2"	3"	13/16"	Left end	Top flg
Web	9'-5 1/2"	6"	13/16"	Left end	Top flg
Web	9'-11"	3"	13/16"	Left end	Top flg
Web	9'-11"	6"	13/16"	Left end	Top flg
Web	21'-6 1/4"	9"	13/16"	Left end	Top flg
Web	21'-6 1/4"	3"	13/16"	Left end	Top flg
Web	21'-6 1/4"	6"	13/16"	Left end	Top flg
Web	21'-9 1/4"	6"	13/16"	Left end	Top flg
Web	21'-9 1/4"	9"	13/16"	Left end	Top flg
Web	21'-9 1/4"	3"	13/16"	Left end	Top flg

----- Burns ----------

Type	End	Surface	Length X	Depth Y	Setback	Side	Angle	Notch D
Bevel cut	Left	Bottom Flange	2 ¼	-------	0	----	-45	-------
Rat Hole	Left	Bottom Flange	3 ¾	1 7/8	0	----	-------	-------
Bevel cut	Left	Top Flange	2 ¼	-------	0	----	45	-------
Rat Hole	Left	Top Flange	3	1 7/8	0	----	-------	-------
Block	Right	Bottom Flange	2 ½	1 1/8	0	----	-------	-------
Block	Right	Top Flange	2 ½	1 1/8	0	----	-------	-------

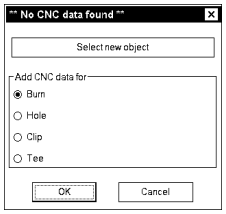
Side:

NS = Near side — Left = Left side of center line in column web gage.

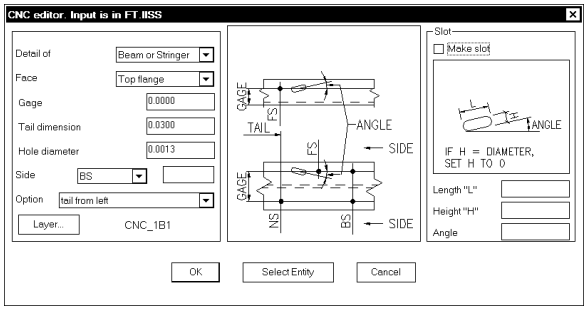
FS = Far side — Right = Right side of center line in column web gage.

BS = Both side

EDIT Data

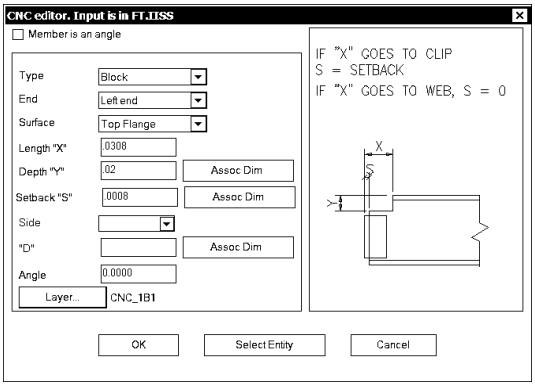


This program is for editing, adding or viewing all of the xdata of a single hole, clip, tee or burn. The dialog box shown below will open if a hole was selected with CNC xdata.



Holes: A slide drawing of the current selection is shown for reference. The value for "Detail of" can be a “Angle", "Beam or Stringer” or "Column or Brace". "Column or Brace" includes everything where the gage of the web is measured about the center line. The "Side" value is not important if the shape is a channel. You can change the layer the hole is on with the “Layer” button. The hole must be on the layer with the same name as the main mark the hole is in. The hole can be changed to a slot by

putting a check in the box for “Make slot” and filling in the values or a slot can be changed to a hole by unchecking “Make slot”. To view the values of another hole, pick "Select Entity". You can use crossing, window or a point to select.



Burns: If you are adding xdata for a burn such as a block, select the length dimension of the block and the box on the right will be displayed with the selected value in the Length “X” box. Pick “Assoc dim” next to Depth “Y” and select the block depth dimension. This will put the value in the Depth “Y” box and associate the two dimensions with the xdata. If you use Replace Dimension to change the depth later, the depth value will be updated in the burn xdata for the block.

Side and Angle: Side is used for flange notch, flange thinning and skewed cuts as listed below. Angle can be 0, positive or negative. The sign of the angle determines which way the cut goes for a skewed cut. Refer to the slide in the dialog box to see which way positive and negative angles are cut.

Skewed cut on web, top or bottom flange or angle leg: There are several variations for X, Y and Angle that are used to determine the size and direction of this cut. In all variations the sign of the angle determines the location of the cut not the side. See the slides in the program. The side option is only important as noted below. The variations are as follows:

- X= 0, Y = 0. (Flange cut only). If side is BS, Y is full width of flange. If side is NS or FS, Y is half width of flange. Y and angle are used to calculate X.
- X= 0, Y = 0. (Web cut only). Y is full depth of web. Y and angle are used to calculate X. The side value determines which web gets cut for tubes only.
- X > 0, Y = 0. (Flange cut only). If side is BS, Y is full width of flange. If side is NS or FS, Y is half width of flange. X and Y are used to calculate angle. Angle may be 0 or -1.
- X > 0, Y = 0. (Web cut only). Y is full depth of web. X and Y are used to calculate angle. The side value determines which web gets cut for tubes only. Angle may be 0 or -1.
- X = 0, Y > 0. Angle and Y are used to calculate X. See note 1 below.
- X > 0, Y > 0. X and Y are used to calculate the angle. Angle may be 0 or -1 to determine direction of cut. See note 1 below.

Note 1: If side = NS or FS there is only one cut. If side = BS there are two cuts. If it is BS of a flange, both corners of the flange are cut and Y must less than or equal to the flange width. If it is BS of a web, both webs are cut on one corner for tubes only.

A 45 degree full width cut of a flange can be entered as:

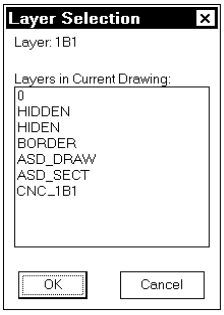
- X = 0, Y = 0, Side = BS, Angle = 45 or -45. (This is the most common way.)
- X = flange width, Y = 0, Side = BS, Angle = 0 or -1.
- X = flange width, Y = flange width, Angle = 0 or -1.
- X = 0, Y = flange width, Angle = 45 or -45.

Y must be greater than 0 for a chamfer cut.

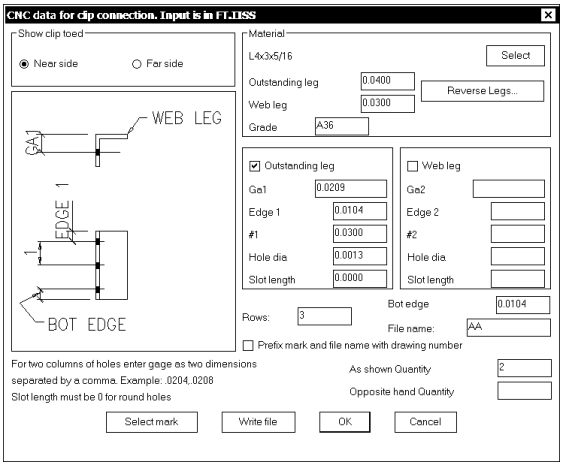
A 45 degree chamfer cut of a flange can be entered as:

- X > 0, Y = X, Angle = 0 or -1. NS or FS cuts one corner. BS cuts both corners.
- X = 0, Y > 0, Angle = 45 or -45. NS or FS cuts one corner. BS cuts both corners.

Flange Thinning: If the member is not an angle and the angle of the flange thinning is not 0 and the flange is thinned on both sides the end of the member is automatically skewed.
Angle, Channel, Other: These options change the reference slide and some of the options to make it easier to understand how each input is used for different materials. The option selected is not saved as part of the burn data.

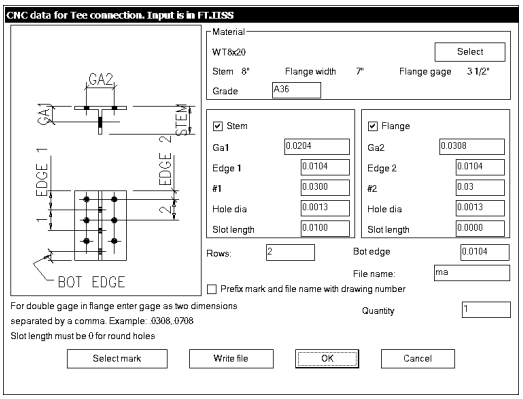


Selecting the Layer button pops up a list box, (shown to the left), of all existing layers in the drawing. Select the layer with the same name as the main mark and pick OK. All CNC holes and burns MUST be on a layer that has the same name as the main mark of the piece the hole or burn is in. This layer is created automatically when a new piece is detailed. If you copy an existing piece to make a new one you MUST create a new layer with the same name as the new main mark and change all entities of the new piece to that layer. The Change Layer command does all of this for you.

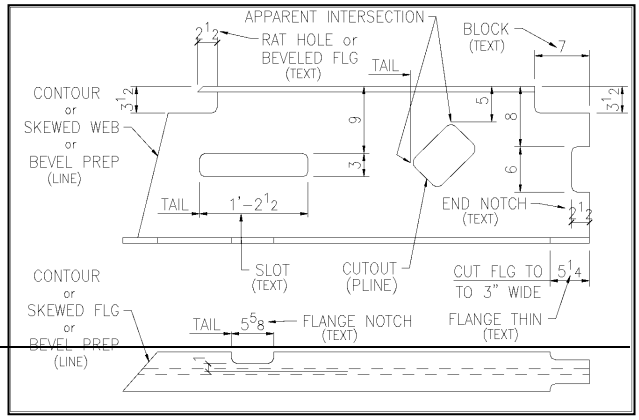


The dialog box shown below will open if a mark was selected with CNC xdata for clip angles.

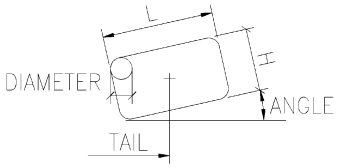
For reference, the "web leg" is the leg shown. If holes are in both legs then both "Outstanding leg" and "Web leg" check boxes must be checked. If you uncheck a box the xdata for the holes for that leg will be removed.
Select mark: saves the xdata to the current mark and lets you select another mark.
Write file: creates a DSTV file. If the file exist you are given the option to replace the file or add the quantity to the existing file.



OK: saves the xdata to the current mark and closes the program.
Cancel: cancels all changes and closes the program.
The dialog box shown to the left will open if a mark was selected with CNC xdata for split tees. It functions in the same way as the dialog box for clip angles above.



The drawing to the right shows the entity that each burn is saved in.



A rectangular slot is shown to the left. The Hole Diameter is used for the radius of the corners. A 1” hole diameter equals 1/2” corner radius.

The drawing below and on the next page shows all of the burn types and conditions.

Note:

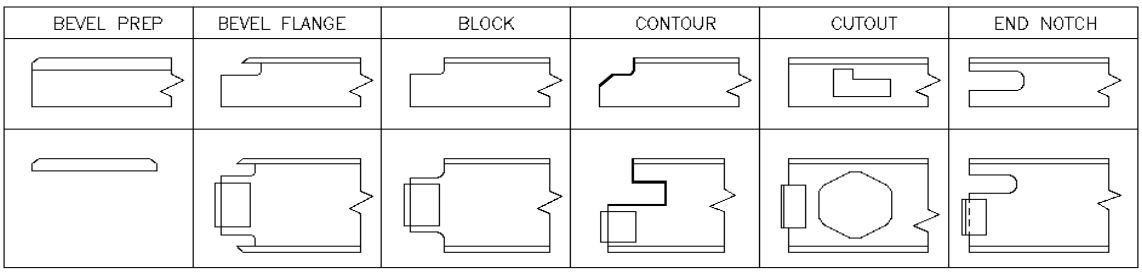
- A notch can have an angle other than 0 making it skewed.
- Any corner of a beam web can have only one of the following: Block, Flange Thin, Rat Hole or Beveled Flange.
- There should be only one contour per end per web and flange but can be used with other burns. Because CNC data for Contours and Cutouts is described graphically by the line with a base angle of 0 degrees, you must enter an angle of 90 when the burn is on a column that is detailed standing up. If the burn is on a sloping beam or brace detailed in position the angle is the angle measured from a horizontal line.
- The “X“ and “Y“ values that locate a cutout go to the left most point for “X” and top most point for “Y”. This is either a line or the intersection of two lines. If it is an intersection and the intersection is rounded then it would be to the apparent intersection of the two lines.
- All angle shapes must be detailed with the outstanding leg on top or on the left side when using contours or cutouts.

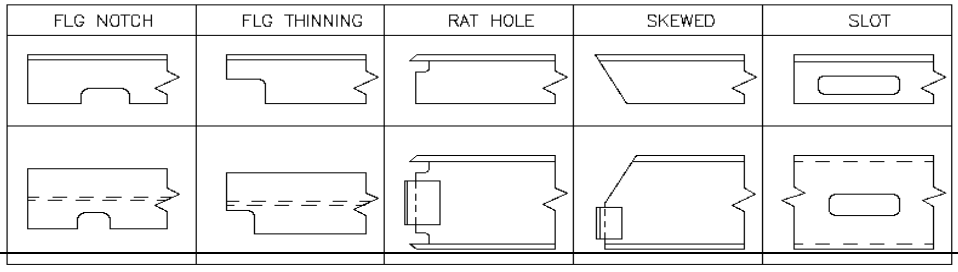
CHOOSE ONLY ONE FROM EACH LINE FOR MAKING COMBINATIONS AT THE ENDS OF FLG AND WEB.

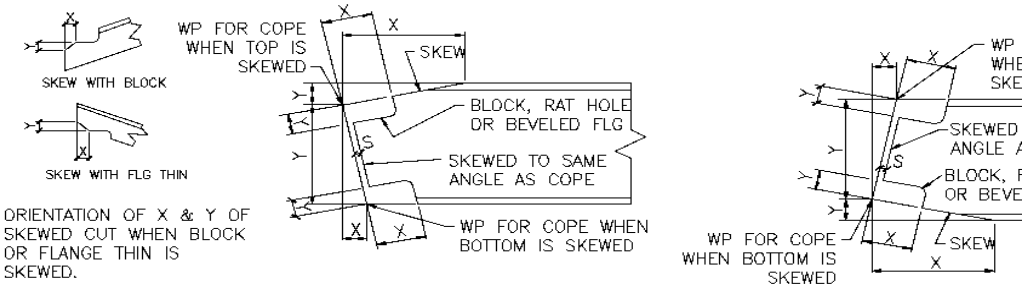
--- END OF WEB BURNS ---
1) BEVEL PREP
2) BLOCK, BEVELED FLG, RAT HOLE
3) CONTOUR, NOTCH
4) SKEWED
--- INTERIOR WEB BURNS ---
1) CUTOUT
2) SLOT

--- END OF FLG BURNS ---
1) BEVEL PREP
2) FLG THIN
3) CONTOUR, NOTCH
4) SKEWED
5) SLOT
--- INTERIOR FLG BURNS ---
1) CUTOUT
2) NOTCH
3) SLOT

DO NOT COMBINE END CONTOUR WITH NOTCH.
A SKEW CAN NOT CROSS A CONTOUR. A SKEW MAY CROSS A BLOCK, BEVELED FLG, RAT HOLE, FLG THIN OR NOTCH.
IF THE ANGLE OF A COPE OR FLG THIN IS NOT 0, THE END OF THE WEB OR FLG MUST HAVE A SKEW CUT TO THE SAME ANGLE.
ONE PER END PER SURFACE – BLOCK, FLG THIN, RAT HOLE, BEVEL PREP, BEVELED FLG, END NOTCH, CONTOUR AND SLOT.
TWO PER END PER SURFACE – SKEWED.
MULTIPLE PER SURFACE – CUTOUT, FLG NOTCH.







The drawing below right shows the many combinations you can have for skewed beam ends. The xdata for the blocks is stored in the "X" value. The xdata for skewed web burns is stored in the line where noted in the drawing.

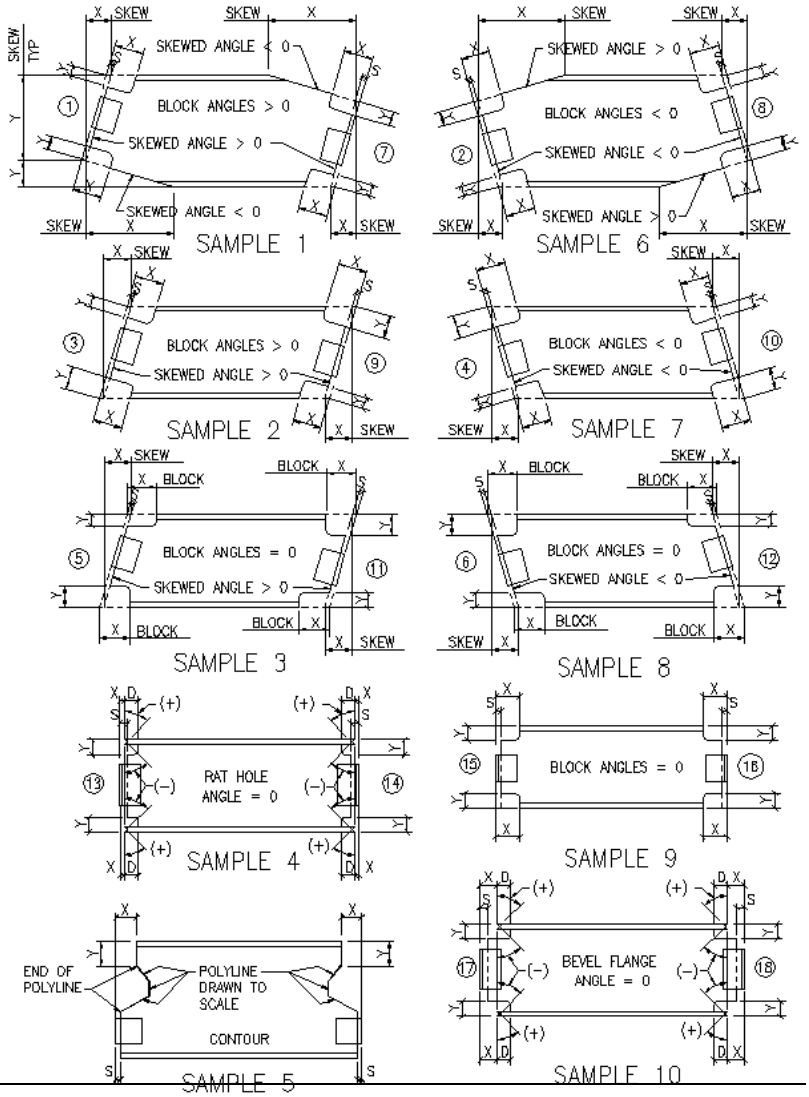
Since one entity can represent only one burn type, if you have a skewed web or flange and the web or flange is also beveled you will need to pick a different entity to add the Bevel prep data to. It can be any entity as long as it is on the correct layer.

EDIT DSTV file

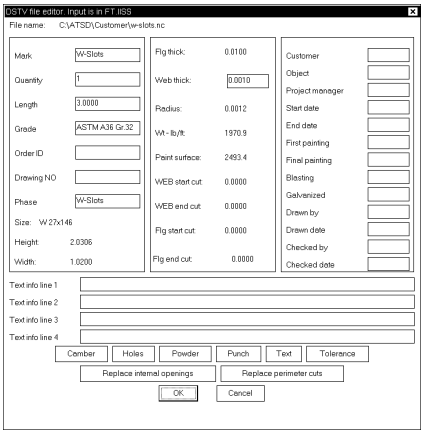
Note: You can edit most aspects of the file by changing values in the editor but if you want to add or change something not available in the editor you must draw the views first, edit the drawing manually that needs changing and convert the perimeter of the view to a polyline before running the DSTV file editor.

The first prompt asks if you want to "Draw data from file <N>?" If you answer Yes the program will draw the views from the selected file using the program to graphically check DSTV files and then opens the DSTV file editor.

The main dialog box of the DSTV editor opens and is shown below. You can edit any of the active edit boxes



shown. The other information is shown for reference only. You can change the length of the member but not the member size. The items that are blank in the example are reference information and are usually not needed.



Camber: Opens the dialog box to edit the camber. You can have camber on any of the four faces. To add a camber, pick the Add button. Enter an X location, (the distance from the left most end of the member to the point of camber), and an amount of camper at that point and pick OK. Remove a camber by selecting one from the list labeled "X location" and pick "Remove". Changing the face will refresh the "X location" list to show the camber for that face. Pick OK to save your changes and close the box. Pick Cancel to ignore your changes and close the box.

Holes, Powder, Punch and Text: All of these works similar to Camber. To edit an existing value, select the item to change from the X–Y location list on the left side and enter a new value on the right side. Pick the Add button to add an item. The defaults are from the item selected. Pick OK to add the item, pick cancel to void the change.

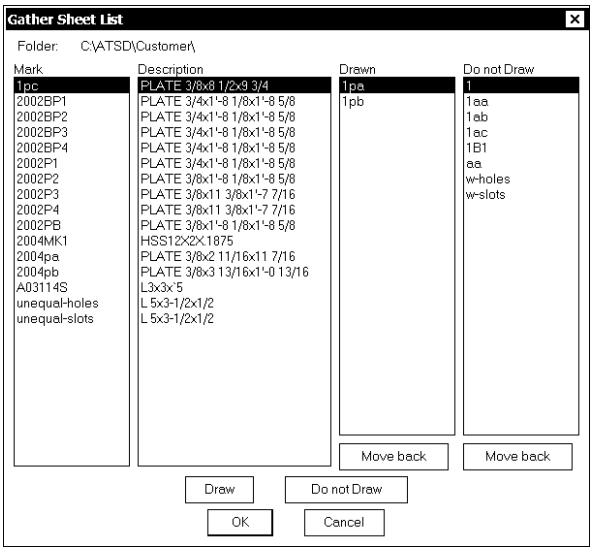
Replace all holes: This button is in the Holes dialog box. You can replace all holes on one or multiple faces by selecting the drawing that was created by drawing the data from the file. This drawing is drawn to scale and all editing to it must be to scale. If you change any holes, the location and size of the hole must be to scale. When selecting the drawing, all entities of all faces of the drawing should be selected including the text under each face. If you only need to replace the holes in the top face, for example, you could select just that face and holes and text that says TOP VIEW only if the left most end of the top face is at an "X" value of 0. That is, no other face extends to the left of the left end of the top face. The program will assume the "X" value of the left end to be 0 for calculating the "X" location of the holes. Replacing all holes will change the "Ref Y from" value for all holes.

Tolerance: This lest you add a minimum and maximum tolerance value to the member. You can remove a tolerance by removing one or both values and pick OK.

Replace internal openings: This is for adding, removing or changing internal opening in a face. The "hole" must be one polyline and the size and location must be to scale. The change is made on the drawing that was created by drawing the data from the file. See "Replace all holes" above for selecting all objects.

Replace perimeter cuts: This is for adding, removing or changing external cuts, blocks or copes in a face. The entire perimeter of the face must be one polyline and must be to scale. The change is made on the drawing that was created by drawing the data from the file. See "Replace all holes" above for selecting all objects.

Gather marks



Use this program to draw a piece from its DSTV file. The DSTV files will be read from the folder of the current open drawing. The dialog box shown to the left will be refreshed each time the program is launched but will remember if a file has been drawn or put in the "Do not Draw" column. Any new files will be added. Any files deleted will be removed. If you do not want to draw a piece, select it and pick the "Do not Draw" button. To draw a piece, select it and pick the "Draw" button. The box will go away and you will be prompted for an insertion point. After the piece is drawn the box will open back up and the mark that was drawn will be moved to the "Drawn" column. If you want to draw the same piece again, select it from the Drawn column and pick the "Move back" button. Now you can select it from the "Mark" column and pick the "Draw" button.

Write DSTV for

Note: All DSTV files will be saved in the folder of the drawing the file was made from.

All Drawings: This will create a DSTV file for each piece on all drawings in a selected folder. It will not create files for plates or clips.

Current Drawing: This will create a DSTV file for each piece on the current drawing. It will not create files for plates or clips.

Single object: This will create a DSTV file of the details selected. Select the main mark under the detail for beams, columns and single member bracing. Select the sub mark for stair stringers and double angle bracing. It will not create files for plates or clips.

Base/Cap plates: This will open the column base/cap plate editor. Select a mark or create a plate and pick OK. A DSTV file will be created for the plate selected.

Write DSTV file for Clips/Tees Angles

Note: All DSTV files will be saved in the folder of the drawing the file was made from.
Clip angles include end clips on beam web and flanges, on gusset plates, on stairs and on columns.

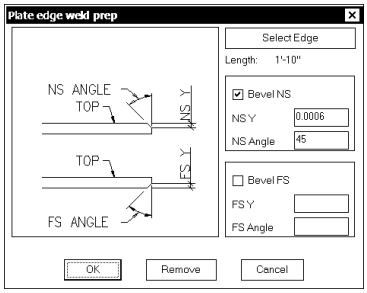
All Drawings: This will create a DSTV file for each clip angle or tee on all drawings in a selected folder. If a mark is used more than one time the quantity will be increased for each mark found. Because of this increase in quantity you should not run this more than once on a set of drawings. If you must repeat the command you should delete all existing *.NC files from the folder or copy the drawings to a temp folder and run the program on that folder.

Current Drawing: This will create a DSTV file for each clip angle or tee on the current drawing. If a mark is used more than one time the quantity will be increased for each mark found.

Single mark: Select a clip angle or tee mark and a dialog box will open showing the CNC data for the clip/tee. If no CNC data is saved with the mark you have the option of adding the data to it. Pick “Write file” and a DSTV file will be created for the clip/tee. Pick “Select Mark” to select another clip/tee. Do not include the ”R” or “L” in the mark if the clip/tee are right & left. It will be added when the file is written. Pick “OK” to close the box. Any changes made will be saved to the clip/tee mark. Pick Cancel to close the box without saving any changes.

Write DSTV file for Plates

The program will ask you to select the plate mark or press Enter to input the mark and plate size. The mark may be part of the description text but is not required to be. Example `P`6x10`8x1’-3`8 pa. If it is part of the description it should be at the end of the text. This is typically how the gusset plate program labels a plate. If a description is not selected you will be asked for the plate width, length and thickness. For the width and length you can enter a numerical value or the letter “S” to select two points for determining the width and length. Next you are asked for the grade of steel and quantity and then to verify the file name for the CNC data. The next step is the select the plate and all holes and cutouts in the plate. The last step is to add a bevel edge prep for weld if needed.

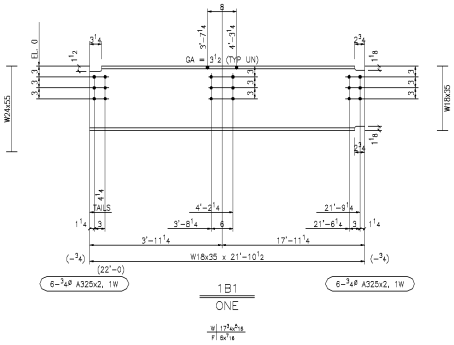


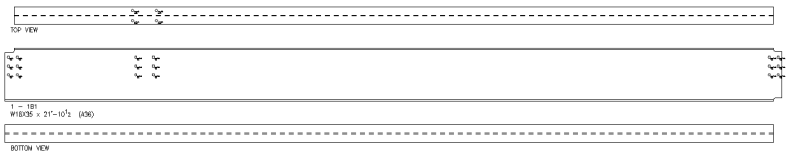
- Plates and all cutouts in plates must be made with closed polylines.
- Round holes must be made with circles and can be concentric circles. The outer circle establishes the diameter.
- Slotted holes must be made with a closed polyline. Multiple polylines inside each other to “fill in” the slot must be at the same centers.
- All details of the plate must be drawn to scale and to the scale of the drawing (no change scale allowed).
- Gusset plates and their clips drawn by the brace program may be used to create DSTV files.
- The gusset plate and holes may be selected using a window to select all objects at once.
- CNC does not need to be turned on to use this program.
- Plates cannot be “batch” processed. They must be processed manually one at a time.

The DSTV files are saved to a file with the same name as the piece mark. The file name extension for DSTV files is set in your customer detail configuration. These files will be saved in the same folder the drawings are in. If a member with CNC data has a sub piece mark such as a stair or frame or if the main mark begins with a letter the DSTV file created will be the piece mark prefixed with the drawing number. For the piece mark ma on sheet 43, the DSTV file name will be 43ma. If the mark is a right and left two DSTV files will be created, 43maR and 43maL.

Graphically Check DSTV file

This will read a selected DSTV file and draw the web, top, bottom, and rear view if required, of the member to scale showing all holes and burns. Channels will be drawn toed near side and upside down.



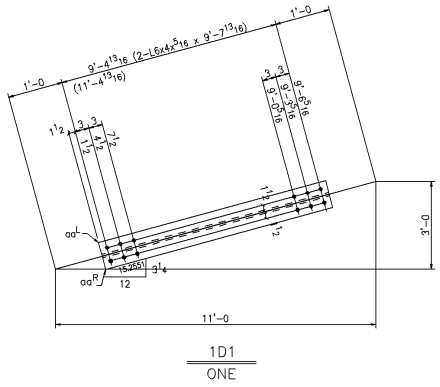


Holes will be labeled with an X and Y dimension. X is the distance from the center of the hole to the left end. Y is the distance from the reference axis. For beam web holes this is either the top or bottom flange. Beam, Column and Brace holes in the flange is the flange centerline. It is the heel for angles. Edges that are beveled will be drawn thicker.

SUB MK

SUB MARK

This command is for labeling a sub piece with a mark attached to a leader. This command does everything that the command *Change CNC Layer* does. After the mark is placed you are asked for the quantity, member size, cut length and grade. Once these items are entered you select all entities that are associated with this mark. A layer is created with the same name as the sub mark prefixed with "CNC_" and all entities selected are moved to this layer. The only entities that must be on the new layer are the holes, text and lines with the CNC data.



In the example shown to the left the sub mark aa^R contains the CNC data for the quantities, length, size and grade. The detail is on layer CNC_aa. The holes in aa^L do not have any CNC data stored in them.

Tips:

Use AutoSD > CNC > Graphically Check DSTV file to redraw the piece from the DSTV file to see if it looks right. This is the file the piece will be fabricated from so it has to be right and you will need to give the file to the fabricator. The file name is usually the same as the piece mark and will be in the same folder as the drawing file. If the piece does not look right after drawing it with “Graphically Check DSTV” it may be easier to fix the file using AutoSD > CNC > Edit DSTV file.

To fix the detail that was drawn using “Graphically Check DSTV”, all changes must be to scale. The “X” and “Y” values noting the location of holes is for visual reference only. The text is not used by the DSTV editor to locate the holes so changing the “X” or “Y” value will have no effect. The size and location of each hole must be to scale.

There is no CNC data associated with plates. The DSTV file information for plates is based on the scale of the plate. It must be to the scale of the drawing, not enlarged, and must be made with a closed polyline. All holes in the plate must be to scale. Round holes may be made with circles or donuts. Slots and other cutouts must be made with a closed polyline.

The Member size, quantity, length and grade are stored in the main mark. If you make a change to one of these on the detail you will also need to make the same change in the main mark.

If you have two columns where the only difference is the orientation requiring two different marks, an easy way of creating the second DSTV file is to copy the first one, rename it the mark of the second column and use the AutoSD > CNC > Edit DSTV file to change the mark in the file.

Appendix A - Installation

This section provides the information required to install AutoSD Steel Detailing on your computer system.

Your AutoSD software comes to you with the following items:

1 - AutoSD Steel Detailing User Manual
1 - Keyboard text location card
1 - CD
1 - Hardware lock

SentinelPro Drivers for Hardware Lock

The drivers for the hardware lock are installed by the installation program. If you need to change the configuration or reinstall the driver, run the program in the Sentinel folder of the CD.

INSTALLING AutoSD STEEL DETAILING

Installing your AutoSD Steel Detailing software is simple. Just run the installation program. All the necessary files will be copied to the hard disk.

The AutoCAD or BricsCAD program should be installed first on the hard disk. After AutoCAD or BricsCAD has been installed, follow these procedures.

Windows 2000, NT, XP, Vista, Windows 7, 8

Put the CD-ROM in your CD-ROM drive.
Close all open applications.
From the Task Bar, select Start/RUN.
Type “D:\autosd_setup” where “D” is your CD-ROM drive.
Press enter.
Follow the on-screen instructions.

When the installation is complete you will have to configure AutoCAD or BricsCAD.

The following is not available with AutoCAD LT.
Since the release of AutoSD 2012, you can run multiple versions of AutoSD at the same time by installing each version of AutoSD in a separate folder and creating a profile in AutoCAD for each version of AutoSD. AutoSD 2012 and later can run with one older version and multiple newer versions.

If you're current version of AutoSD is version 2012 or later and you want to also run AutoSD 2015, simply install AutoSD 2015 in a folder with a different name from where the earlier version of AutoSD is installed.

Example:
AutoSD 2012 is installed in C:\autosd
Install AutoSD 2015 in C:\autosd 2015

Note: You can only use Windows "Uninstall Programs" function to uninstall the last version of AutoSD that was installed. In the example above, you would not be able to uninstall AutoSD 2012 because AutoSD 2015 was installed after AutoSD 2012. You would have to delete the AutoSD 2012 folder manually to remove the installation.

Open a drawing in AutoCAD and pick Tools > Options > Profiles
If you do not have a profile named for the current version of AutoSD, pick rename and change the name of the current profile to AutoSD 2012.

To make a new profile, pick "Add to list". Enter the name "AutoSD 2015" and pick "Apply & close".
Pick "Set Current"
Pick the Files tab
Change the "Support file search path" for AutoSD to C:\autosd 2015\support
Pick Apply
Pick OK
Load the AutoSD menu as noted below.

To change from one version of AutoSD to another just pick Tools > Options > Profiles, select the profile you want and set it current.

AutoCAD Release 14 configuration

From the pulldown menu select **TOOLS** and then **Preferences** or type **CONFIG** and press enter**.** A dialog box is displayed.

Select Files
Double click on Support file search path.
Select Add. Add the following line in the newly created box.

C:\AUTOSD\SUPPORT

Where:

C:	is the drive where AutoSD is installed.
AUTOSD	is the name you gave for the AutoSD folder.

Close AutoCAD
Start AutoCAD To enable the new search path.
Pick the "Apply" button

AutoCAD Release 2000 - 2013 configuration

From the pulldown menu select **TOOLS** and then **Options** or type **CONFIG** and press enter**.** A dialog box is displayed.

Select Files
Double click on Support file search path.
Select Add. Add the following line in the newly created box.

C:\AUTOSD\SUPPORT

Where:

C:	is the drive where AutoSD is installed.
AUTOSD	is the name you gave for the AutoSD folder.

Pick the "Apply" button

AutoCAD Release 2014 and later configuration

From the pulldown menu select **TOOLS** and then **Options** or type **CONFIG** and press enter**.** A dialog box is displayed.

Select the Files tab
Double click on "Support file search path" or pick the plus sign next to it.
Select "Add" and add the following line in the newly created box.

C:\AUTOSD\SUPPORT

Where:

C:	is the drive where AutoSD is installed.
AUTOSD	is the name you gave for the AutoSD folder.

Pick the "Apply" button

You can add the following to your Trusted Locations to avoid constant warnings about loading files or you can disable the warning feature.

To add folders to "Trusted Locations":
This is for the full version only, not the LT version of AutoCAD.

Double click on "Trusted Locations" or pick the plus sign next to it.
Repeat the following steps for each folder below. If you did not purchase the complete AutoSD Steel Detailing program, some folders will not exist and AutoCAD will warn you of this. In those cases do not include that folder.

Pick the "ADD" button and enter the following line in the newly created box or, (to avoid typo errors), pick the Browse button and select the folder.

C:\AUTOSD
C:\AUTOSD\BM
C:\AUTOSD\BONUS
C:\AUTOSD\BRACE
C:\AUTOSD\CNC
C:\AUTOSD\COLUMN

C:\AUTOSD\DATA
C:\AUTOSD\DATA\BACKUP
C:\AUTOSD\HIP&VAL
C:\AUTOSD\LADDER
C:\AUTOSD\MENUS
C:\AUTOSD\RAILS
C:\AUTOSD\STAIR
C:\AUTOSD\SUPPORT
C:\AUTOSD\WELDS

Pick the “Apply“ button.

To disable the warning feature:
This is for the full version only, not the LT version of AutoCAD.

Select the System tab
Pick the “Executable File Settings” button
Pick the option “Load from all locations without displaying a warning”.
Pick OK
Pick the “Apply“ button.

To disable the warning feature in AutoCAD LT:
Use a Windows registry editor such as regedit.exe. Search for SECURELOAD and change the value to 0

Bricscad 10 and later Pro/Platinum configuration

Type **CONFIG** and press enter**.** A dialog box is displayed.

Expand Program options
Expand Files
Select Support File Search path
Pick the button at the right end of the line with the three dots...
Double click on the blank line at the bottom of the Folder List and then pick the button at the right end of the line with the three dots...
Expand My Computer
Expand Local Disk (C:)
Expand AutoSD
Select Support
Pick OK
Pick OK again

Expand Templates
Select Template
Pick the button at the right end of the line with the three dots...
Change folders to C:\autosd\support and select Autosd.dwt

Pick Open

Select Template path
Pick the button at the right end of the line with the three dots...
Change folders to C:\autosd\support
Pick OK

Loading the menu

AutoCAD:
If you are updating AutoSD 2011 or older version you will need to load the original AutoCAD menu before loading the AutoSD menu.

Pick Tools > Options > Files tab
Expand "Color book Locations"
The second path, (the long one), is the path for the ACAD menu.
Copy this line and paste it in place of the path for Customization Files > Main Customization File.
Change the word color at the end of the line to acad. (ACLT for AutoCAD LT)
Press the enter key after changing the path
Pick the Apply button

Loading the AutoSD menu:
At the command prompt type MENULOAD and press enter.
Pick Browse.
In the "Select Menu File" box pick the arrow at the right end of the "Look in" box and select the drive where AutoSD is installed. In the large box, double click on AUTOSD, double click on MENUS.
AutoCAD 14, 2000 – 2009 and Bricscad: select autosd.**mnu**.
AutoCAD 2010 – 2014: select autosd.**cuix**
AutoCAD 2015 and later: select autosd15.**cuix**
Select Open.

Pick Load	A message box appears warning that loading of a template file overwrites the menu source file.
Select YES	To continue loading the MNU file.

For AutoCAD 14, 2000 - 2005:
Select AUTOSD from "Menu Groups" and pick the Menu Bar tab.
Select where you want to place the AutoSD menu group in the Menu Bar list and pick Insert.
Select Close

All versions of AutoCAD can load the autosd.mnu file. You may need to change "Files of type" to "Legacy menu files (*.mns, *.mnu)" at the bottom of the "Select Customization File"

dialog box to be able to select the AUTOSD.MNU file. The MNU file does not have the ribbon menu in it.
If you have AutoCAD 2010 or later and load the AUTOSD.CUIX menu file you can use the ribbon menu in AutoSD. Use the RIBBON command to turn the ribbon menu on. Use the RIBBONCLOSE command to turn the ribbon menu off. The ribbon menu can be used with or without the pulldown and toolbar menus. Pulldown menus can be turned on and off with the command MENUBAR. Set the value to 1 to turn it on and to 0 to turn it off.

WARNING: If you are updating AutoSD from version 2011 or older, do not use the menus in the sub-folders of C:\autosd\menus. Only load the autosd.mnu, autosd.cuix, autosd15.mnu or autosd15.cuix file with the MENULOAD command. Do not load any auto-sd *.mnu, *.cui or *.cuix files. You should delete these files after transferring any changes you have made to them to the ACAD or a custom menu file.

AutoCAD LT configuration

Using CADBooster:
Using Notepad, copy the contents of C:\AUTOSD\MENUS\LoadLT.lsp
Paste to the end of the file listed below:
For LT2007 - LT2011 (Change 2007 to your version of AutoCAD)
C:\Program Files\cad-port\CADBooster\2007\CADBoosterDoc.lsp
For LT2000 - LT2006 (Change 2000 to your version of AutoCAD)
C:\Program Files\cad-port\CADBooster\2000\LTBoosterDoc.lsp

Using Toolkit Max:
Using Notepad, copy the contents of C:\AUTOSD\MENUS\LoadLT.lsp
Paste to the end of the file listed below:
For LT2002 - LT2010 (Change 2002 to your version of AutoCAD)
C:\Program Files\drcauto\Toolkit\LT2002\Toolkitdoc.lsp

For Cadsta Max:

1. Open a drawing in AutoCAD LT using the CADSTA icon.
2. Click on pull-down menu “CADSTA” > “CADSTA Tools” > “Startup Suite”
3. Click “Startup Suite” tab
4. Click “Add” button
5. Select “C:\AutoSD\Menus\LoadLT.lsp” and click “OK” button to add this to the list.
6. Click “OK” to exit the dialogue box

INSTALLING the HARDWARE LOCK

You must attach the hardware lock to your computer in order to run AutoSD Steel Detailing. If your hardware lock is damaged or faulty, contact AutoSD, Inc. You must return the original to receive a replacement.

The hardware lock affects AutoSD Steel Detailing only. Other programs can run with the hardware lock in place.

Installation of the USB lock:
1. Install the software before installing the USB lock.
2. Connect the USB lock to any open USB port.

Start AutoSD. If you can detail an anchor bolt, you have attached the hardware lock and configured AutoCAD properly.

Starting a NEW drawing

To start a new drawing, start AutoCAD or BricsCAD. If AutoCAD or BricsCAD is already running select Files and New. Select "Use a Template". In the “Select a Template” window select autosd.dwt and pick OPEN. If autosd.dwt does not show in the template list it can be found in C:\autosd\support.

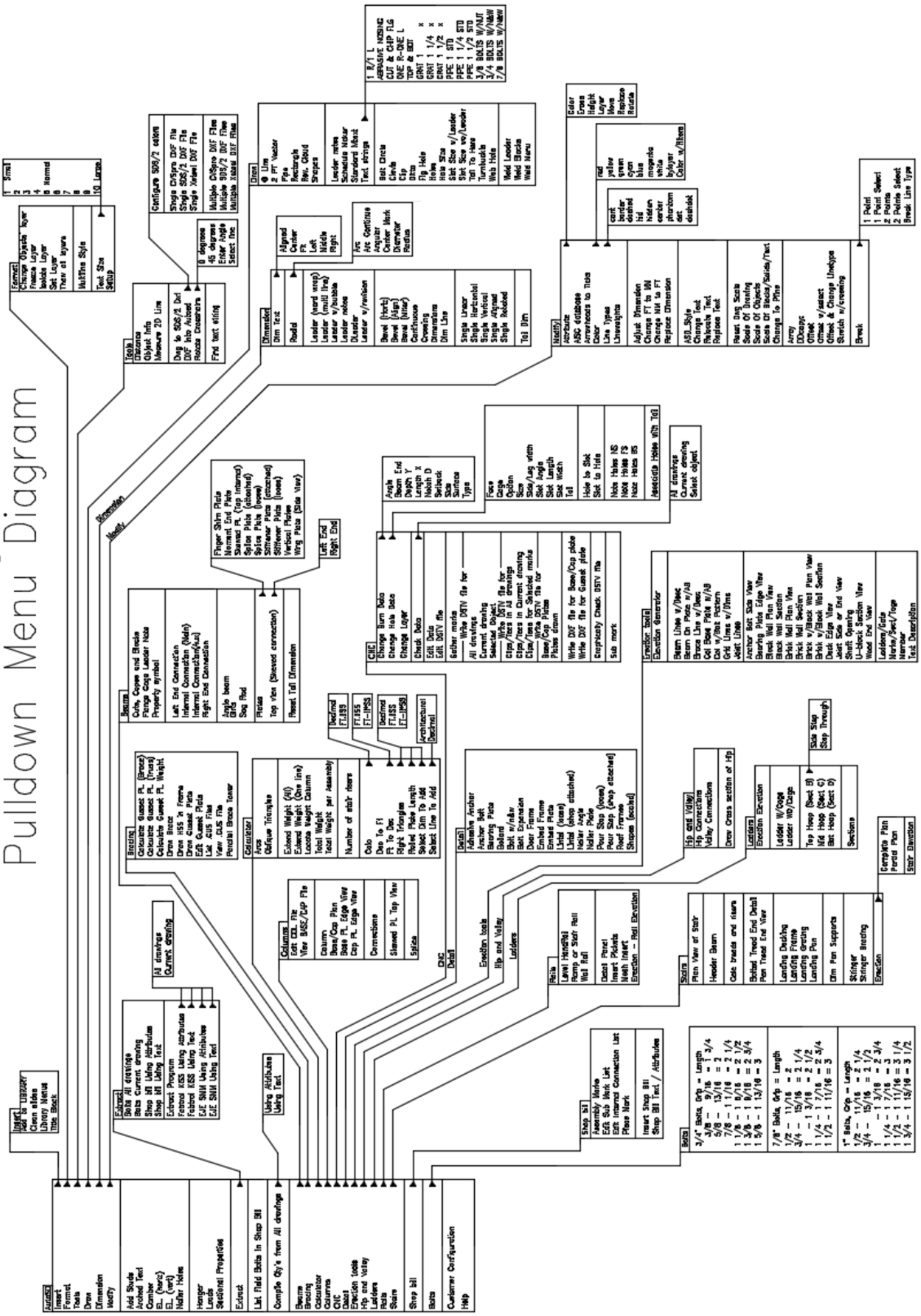
Select OK, Open or Finish.
Run Setup from the AutoSD > Format pulldown menu. A dialog box is displayed to select the units, scale and paper size. You must save the drawing before you can use AutoSD commands. This must be done before you start drawing.

This template file is set up to a scale of 1" = 1', using architectural units on a paper size of 24 x 36. You can create other template files by running SETUP from the AutoSD > Format pulldown menu to set the units, scale and paper size if it is to be different. You can also insert your customer's shop bill and create a template for each customer. This will eliminate the need to run SETUP each time you start a new drawing. Pick "Files "from the pulldown menu and pick "Save As". Select "Save as file type", at the bottom of the dialog box, and select "Drawing template file". Change the file name and folder where it will be saved and pick OK.

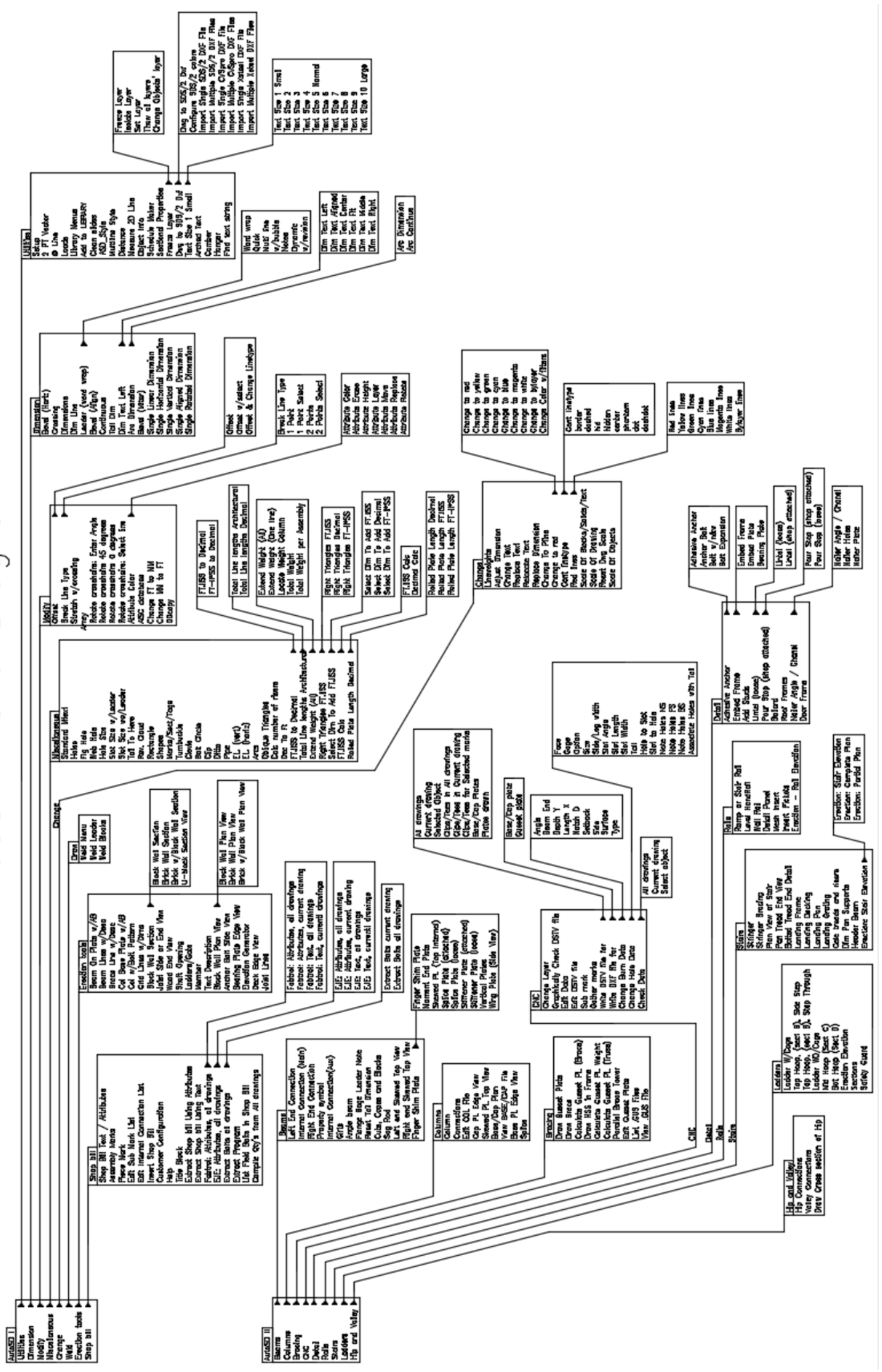
IntelliCAD

IntelliCAD is no longer supported. You should upgrade to AutoCAD 14 or later or Bricscad Pro 10 or later.

There is a drawing called “Pulldown menu diagram” located in your C:\autosd\menus folder. The size is 24" x 36" and the scale is 1" = 1'-0.



There is a drawing called "Ribbon menu diagram" located in your C:\autosd\menus folder. The size is 24" x 36" and the scale is 1" = 1'-0.



Appendix B - Support Files

ASD_CFG.LSP

This file is created when you install AutoSD. It is created in your C:\autosd\support folder. Previous versions created this file on the root of drive C:\. That file is no longer used and may be deleted. Below are the typical contents of C:\autosd\support\asd_cfg.lsp

```
(setvar "MENUECHO" 0)
(setvar "CMDECHO" 0)
(setq asd_drive "C:")          ; The drive AutoSD is on.
(setq asd_dir "/Autosd/")      ; The name of the AutoSD folder.
(setq asd_path "C:/Autosd/")   ; The path of the AutoSD folder.
(princ)
```

If you move AutoSD you will need to edit this file to give the new location of AutoSD.

SHAPE DATABASE

The data base is designed to make it easy to change existing shape properties or add new shapes. The data files are located in your C:\autosd\data folder. The file asd_data.cfg list the names of the files used for each shape. There are three lines for each country of shapes supported. An example of the three lines for US shapes is given below.

Line 1:

```
 ("US")
```

Line 2:

```
(("C"."C")("HP"."W")("M"."W")("MC"."C")("MT"."WT")("S"."S")("ST"."ST")("WT"."WT")
("W"."W")("CF"."CF")("SSC"."CF")("DSC"."CF")("UC"."CF")("Z"."CF")("L"."L")("CB"."W")
("TS"."TS")("PS"."PS")("HSS"."HSS"))
```

Line 3:

```
(("C"."us-c.dat")("HP"."us-hp.dat")("M"."us-m.dat")("MC"."us-mc.dat")("MT"."us-mt.dat")
("S"."us-s.dat")("ST"."us-st.dat")("WT"."us-wt.dat")("CB"."us-cb.dat")("W"."us-w.dat")
("TS"."us-TS.dat")("CF"."us-cf.dat")("SSC"."us-cf.dat")("DSC"."us-cf.dat")
("UC"."us-cf.dat")("Z"."us-cf.dat")("PS"."us-ps.dat")("HSS"."us-hss.dat")
("L"."us-L.dat"))
```

The first line, ("US"), is the country and must not be changed. The second line list the shape designations and their aliases. Example: ("shape" . "aliases"). The third line list the shapes and their corresponding data file. Example: ("shape" . "data file").

The following list the aliases for each shape type.

C = Channels with sloped and parallel flanges
S = Wide flange with sloped flanges

W = Wide flange with parallel flanges
ST = Split Wide flange with sloped flanges
WT = Split Wide flange with parallel flanges
TS = Hollow square or rectangular sections
PS = Hollow round sections
L = Angles, equal and unequal legs
CF = Cold formed light gage "Z" and "C" shapes
TB = Turnbuckle
CL = Clevis
SN = Sleeve nut

The last section of the asd_data.cfg file labeled ("SHAPES"), as shown below, is a list of aliases and corresponding shapes for each country. In each list the first item is the alias and the second item is a list of all shapes associated with that alias. Example for US: ("W" ("HP""M""S""W""CB")), where "W" is the alias and ("HP""M""S""W""CB") is the list of shapes that will be displayed in pop down boxes. "CB" is for castellated beams.

```
("SHAPES")
("listus" ("W" ("HP""M""S""W""CB")) ("CF" ("CF")) ("WT" ("MT""ST""WT")) ("C" ("C""MC"))
("TS" ("TS""HSS"))
("PS" ("HSS""PS")) ("L" ("L")))
("listcan" ("W" ("HP""M""S""W""WWF""WRF")) ("CF" ("CF")) ("WT" ("WWT""MT""ST""WT")) ("C"
("C""MC"))
("TS" ("TS""HSS")) ("PS" ("HSS""DN")) ("L" ("L")))
("listuk" ("W" ("J""UB""UC""UBP")) ("WT" ("BT""CT")) ("C" ("C")) ("TS" ("RHS""SHS"))
("PS" ("CHS""ERW"))
("L" ("L")))
("listeur" ("W" ("DIL""HD""HE""HL""HP""HX""IPE""IPN""IPEA""IPEO""IPER""IPEV")) ("C"
("U""UAP""UPN"))
("TS" ("HSS")) ("PS" ("HSS")) ("L" ("L")))
("listaus" ("W" ("TFB""UB""UC""WB""WC")) ("WT" ("BT""CT")) ("C" ("PFC""TFC")) ("TS"
("RHS"))
("PS" ("CHS")) ("L" ("EA""UA")))
```

Where:
listus is for US.
listcan is for Canada.
listuk is for United Kingdom.
listeur is for European.
listaus is for Australian.

In order to add more members of an existing shape to the data base just edit the file that contains the shape type using the database editor found under AutoSD > Modify > AISC database. As the name implies, it will only edit the shapes used in the US. Shape files for other countries will have to be edited manually.

To add different shape types to the data base, create a file that contains the shape type definitions. Put the file in the autosd\data folder. Edit the file asd_data.cfg and add the new shape and alias, (ie: ("shape" . "alias")) to the second line of the country section. Add the shape and the corresponding data file name to the third line of the country section. Add the new shape designation to the list for it's alias in the SHAPES section.

A copy of the first four lines of the data file us-w.dat is shown below. The format is typical of all “US” data files.

```
 "D" "Wt" "MM D" "MASS" D   WT   AR   tw     Bf    Tf     T      k      r     Ix     Sx     Rx    Iy     Sy     Ry     Zx    Zy   GAGE)
("4" "13" "100" "19" 4.16 13 3.83 0.280 4.060 0.345 2.625 0.750 0.405 11.300 5.460 1.720 3.860 1.900 1.000 6.280 2.920 2.250)
("5" "16" "130" "24" 5.010 16 4.68 0.240 5.000 0.360 3.500 0.750 0.390 21.300 8.510 2.130 7.510 3.000 1.270 9.590 4.570 2.750)
("5" "19" "130" "28" 5.150 19 5.54 0.270 5.030 0.430 3.500 0.813 0.383 26.200 10.20 2.170 9.130 3.630 1.280 11.60 5.530 2.750)
```

The first line is the designation for the dimension in the column below it. This line must be the first line of the file and the order of numbers must be maintained. The remaining lines describe each shape. The entire description for a shape must be on a single line enclosed in parentheses. The numbers are divided into columns to make the file easy to read but in reality there only needs to be one space between each number.

Below is the designation and definition of each column.

"D"	= Nominal depth in inches enclosed in quotes.
"Wt"	= Nominal weight in pounds enclosed in quotes.
"MM D"	= Nominal depth in millimeters enclosed in quotes.
"MASS"	= Nominal mass in kg enclosed in quotes.
D	= Actual depth in inches
WT	= Actual weight in pounds
AR	= Area in square inches
tw	= Web thickness in inches
Bf	= Flange width in inches
Tf	= Flange thickness in inches
T	= Clear distance on web between fillets in inches
k	= Flange thickness plus fillet radius in inches
r	= Fillet radius in inches
Ix	= Moment of inertia about the x axis in inches to the 4th
Sx	= Elastic section modulus about the x axis in inches to the 3rd
Rx	= Radius of gyration about the x axis in inches
Iy	= Moment of inertia about the y axis in inches to the 4th
Sy	= Elastic section modulus about the y axis in inches to the 3rd
Ry	= Radius of gyration about the y axis in inches
Zx	= Plastic section modulus in inches to the 3rd
Zy	= Plastic section modulus in inches to the 3rd
GAGE	= Common bolt gage of the flange.

When adding new sizes or creating new shape files you should keep the sizes in order by depth starting with the smallest depth at the beginning of the file. Also, all sizes of the same nominal depth must be kept together. The shapes will be displayed in the selection dialog box by the order they appear in the data file.

Appendix C - Gusset Plate Details

This appendix provides you with a copy of the slides used in the GUSSET PLATE (BRACE) program. They can be used for reference with the printout of the input and output and by other draftsman to fill out the input form shown on the following page. Once the form is filled out, it can be entered into the computer by your AutoSD operator to run the calculations and get a print of the results which can then be used by the draftsman to make the details. This will speed up the process of detailing braces and gusset plates by draftsman not using a computer, and reduce check time by having accurate details.

GUSSET PLATE INPUT FORM

CONNECTION NO._____ TYPE _____ QUAD _____

LOAD ON BRACE ..... :_____ (May be 0, zero)

MEMBER SIZE ..... :_____ (May be the letter O for Other)

TOTAL AREA OF BRACE(S) ..... :_____ (May be 0, zero)

NUMBER OF BOLTS ..... :_____ (If load and area are 0, zero)

COMPRESSION (-) TENSION (+) BOTH (*)…….. :_____

SMALLER RADIUS OF GYRATION…… :_____ (Input if MEMBER is not given and brace is - or *)

IS BRACE "X" <Y/N>..... :_____ (Input if brace is double angle and is - or *)

BOLT SIZE: 3/4(6) 7/8(7) 1(8)…….. :_____

EDGE DISTANCE..... :_____ (Default is bolt diameter x 2)

BOLT VALUE SINGLE SHEAR..... :_____

GRADE OF STEEL Fy = (36/50)..... :_____

GRADE OF GUSSET PLATE Fy = (36/50) ……. :_____

BOLTS IN DOUBLE SHEAR (Y/N) ..... :_____

(FT.IISS) THICKNESS OF BRACE ..... :_____

(FT.IISS) THICKNESS OF GUSSET PLATE…….. :_____

(FT.IISS) #1 ……..... :_____

(FT.IISS) #2..... :_____

(FT.IISS) INPUT "A" OR <ENTER> TO CALCULATE ……..… :_____ (Input 0, zero, or dimension

HORIZ. CONN IS (0)WELDED (1)CLIP (2)CLIPS (3)END PLATE . :_____ desired for "A" in order to input

(FT.IISS) SETBACK …….:_____ clearance)

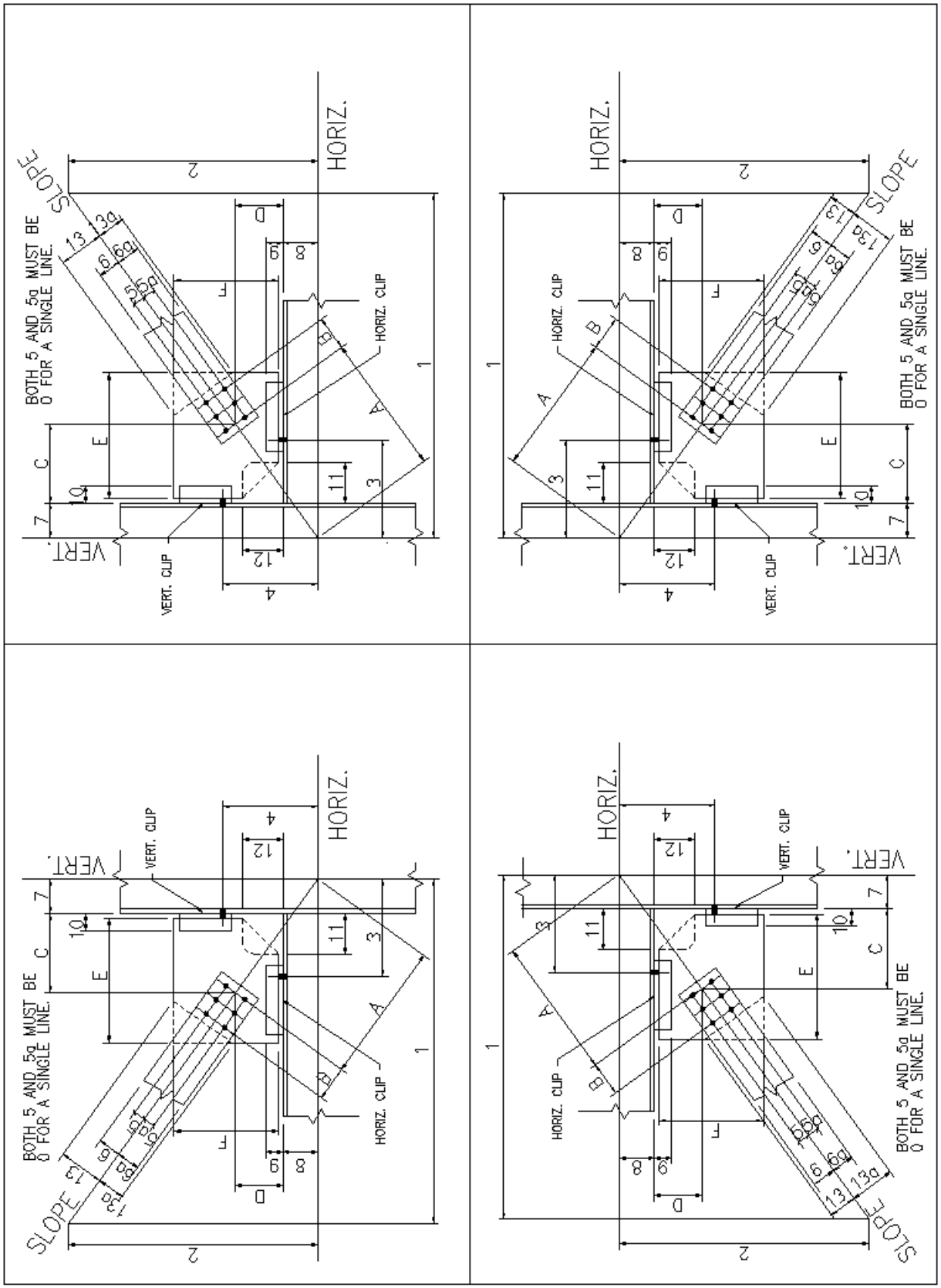
VERT CONN IS (0)WELDED (1)CLIP (2)CLIPS (3)END PLATE …..:_____

(FT.IISS) SETBACK …... :_____

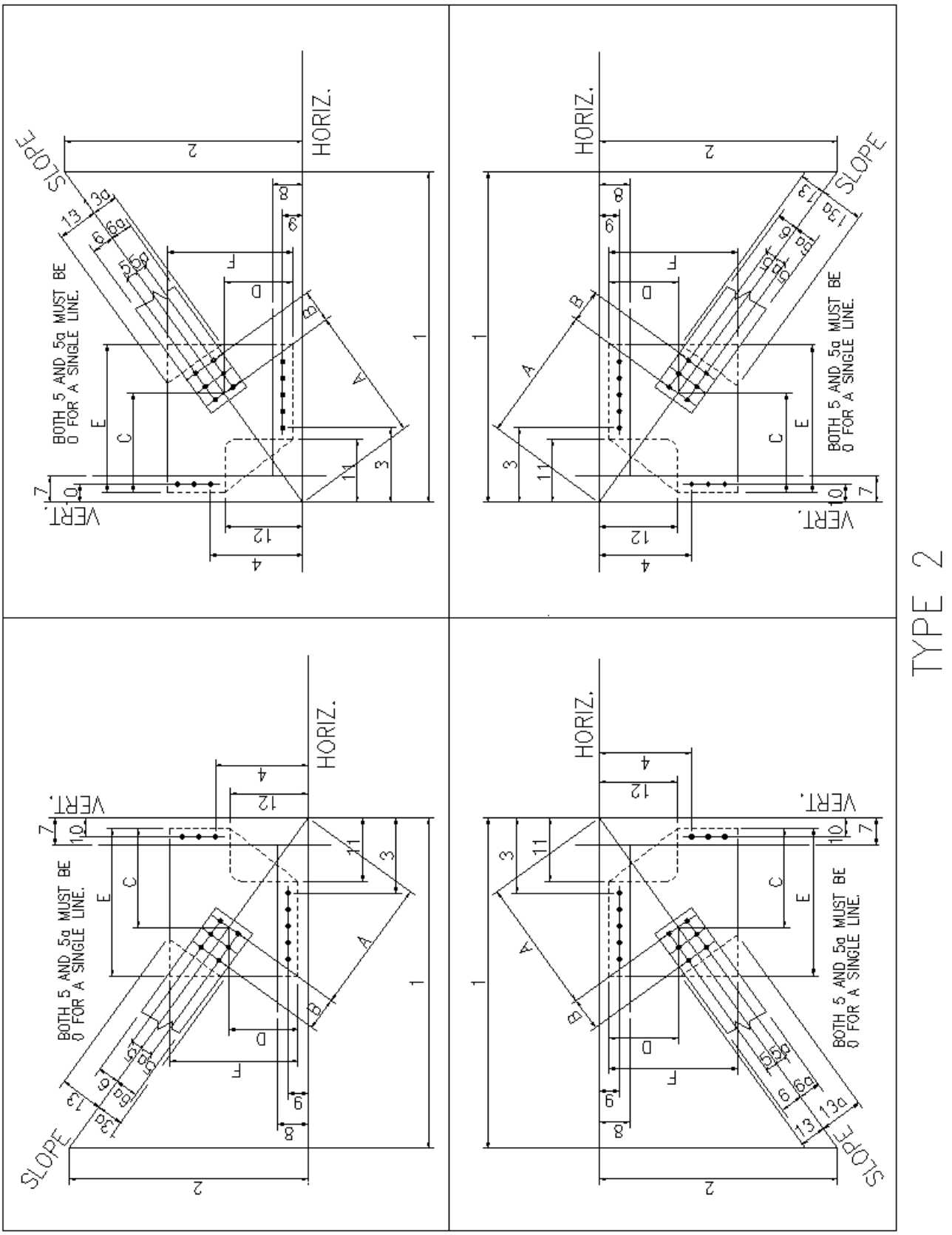
IS CORNER OF PLATE CLIPPED (Y/N) :_____

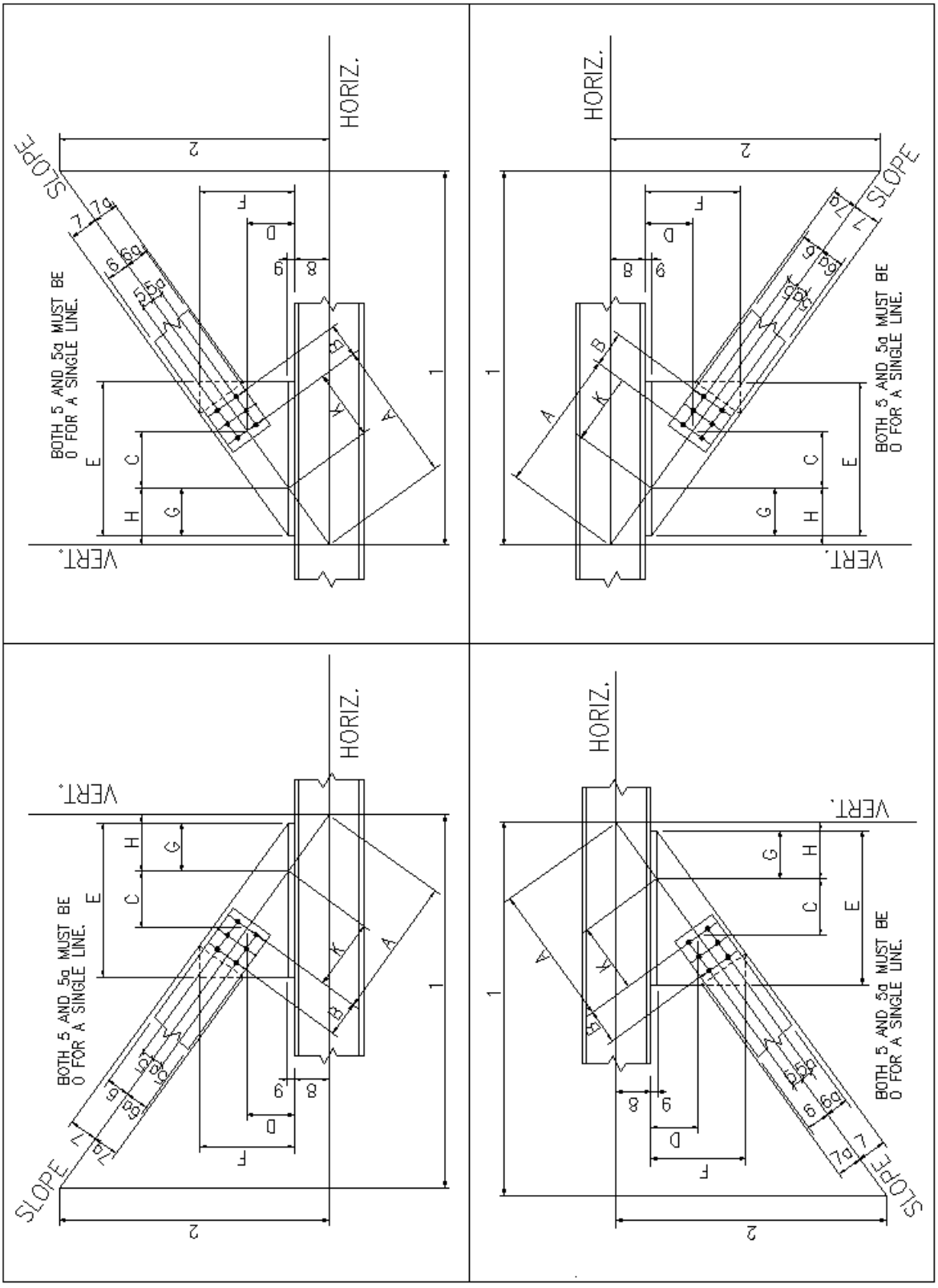
(FT.IISS) MINIMUM CLEARANCE ..... :_____ (If dimension "A" is given)

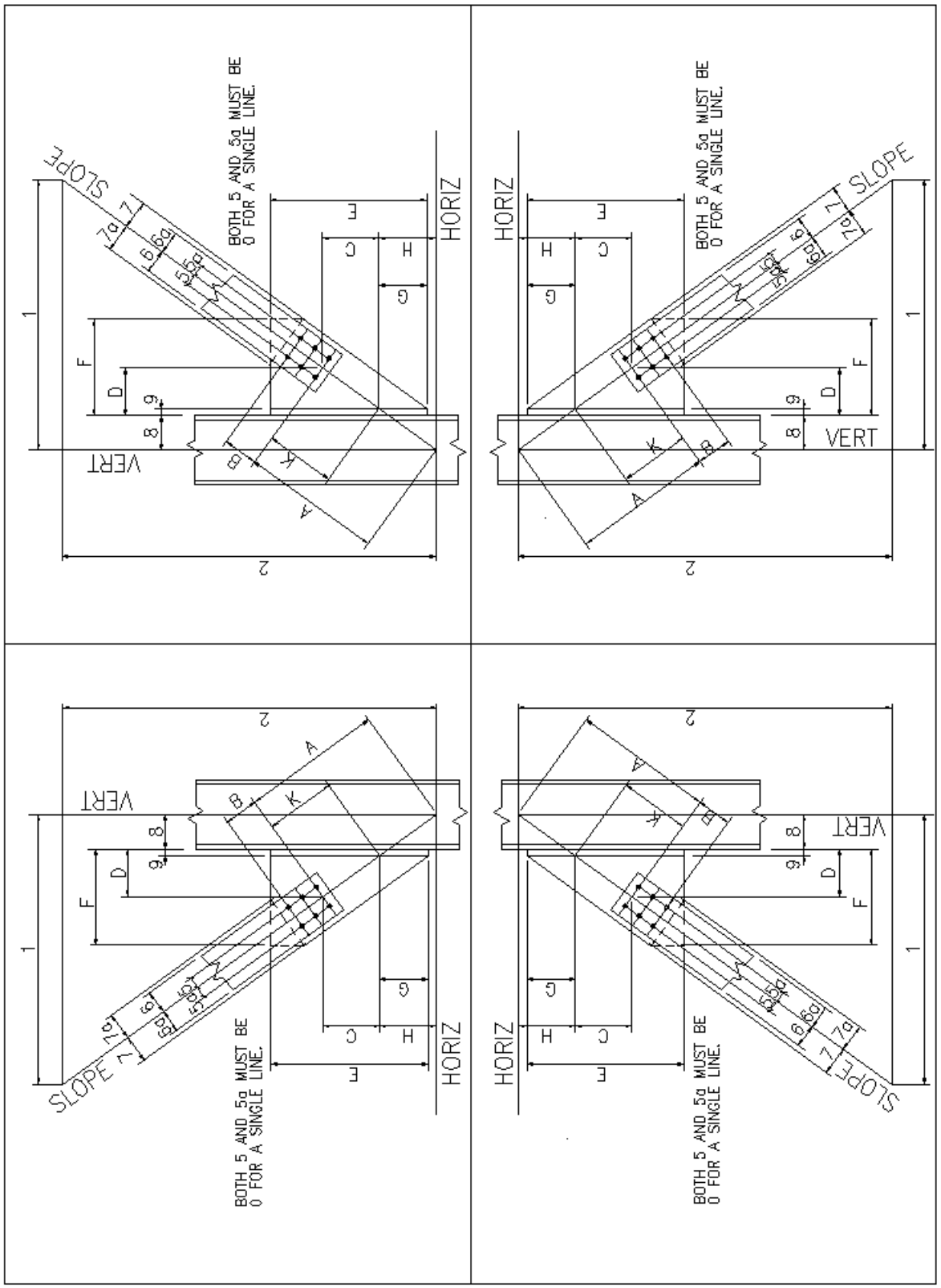
(FT.IISS) #3 …. :_____	(FT.IISS) #6a … :_____	(FT.IISS) #10 ... :_____
(FT.IISS) #4 …. :_____	(FT.IISS) #7 …. :_____	(FT.IISS) #11 … :_____
(FT.IISS) #5 …. :_____	(FT.IISS) #7a … :_____	(FT.IISS) #12 … :_____
(FT.IISS) #5a ... :_____	(FT.IISS) #8 …. :_____	(FT.IISS) #13 … :_____
(FT.IISS) #6 …. :_____	(FT.IISS) #9 …. :_____	(FT.IISS) #13a .. :_____

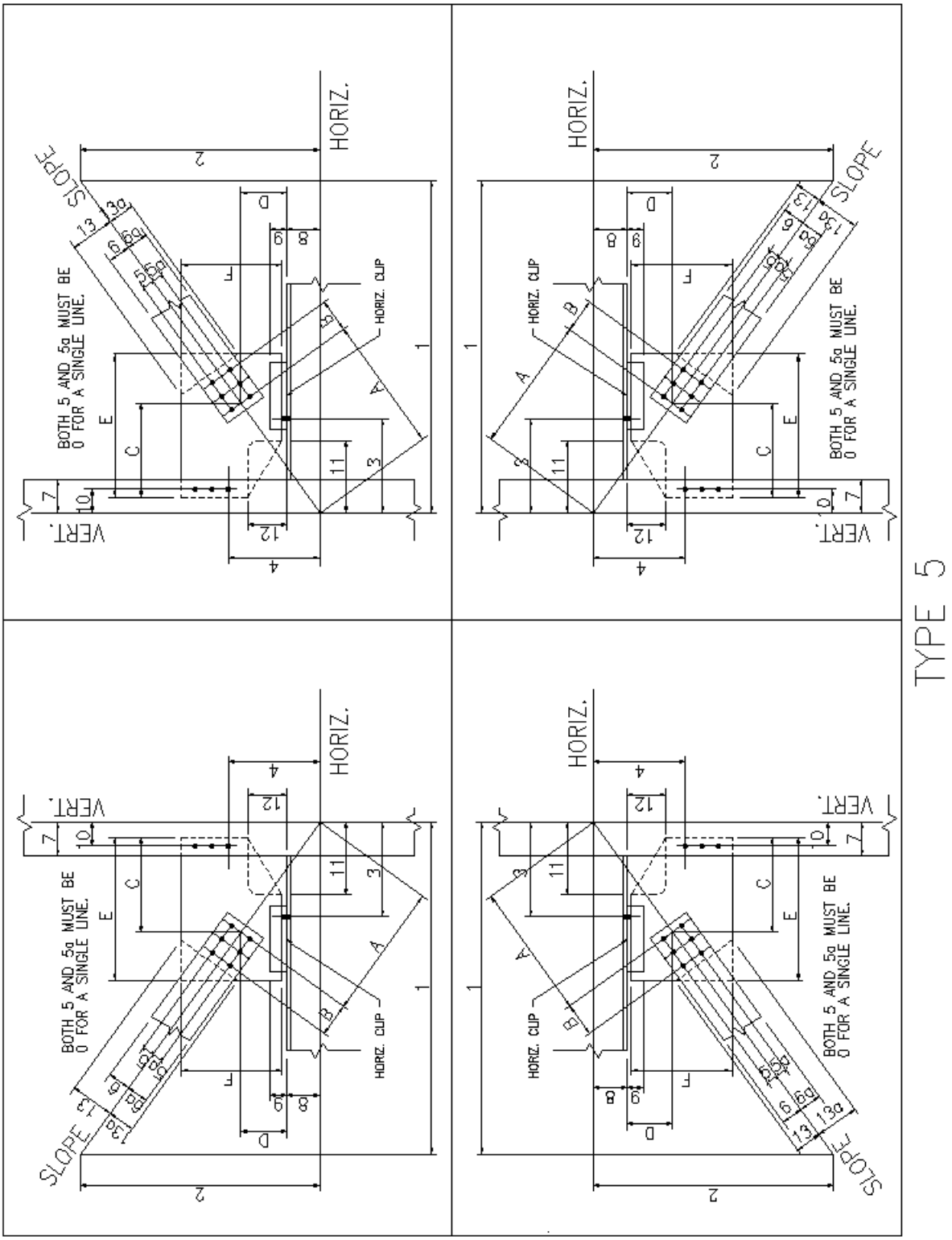

HORIZ.
SLOPE
VERT.
HORIZ. CLIP
VERT. CLIP
BOTH 5 AND 5a MUST BE
0 FOR A SINGLE LINE.

TYPE 1

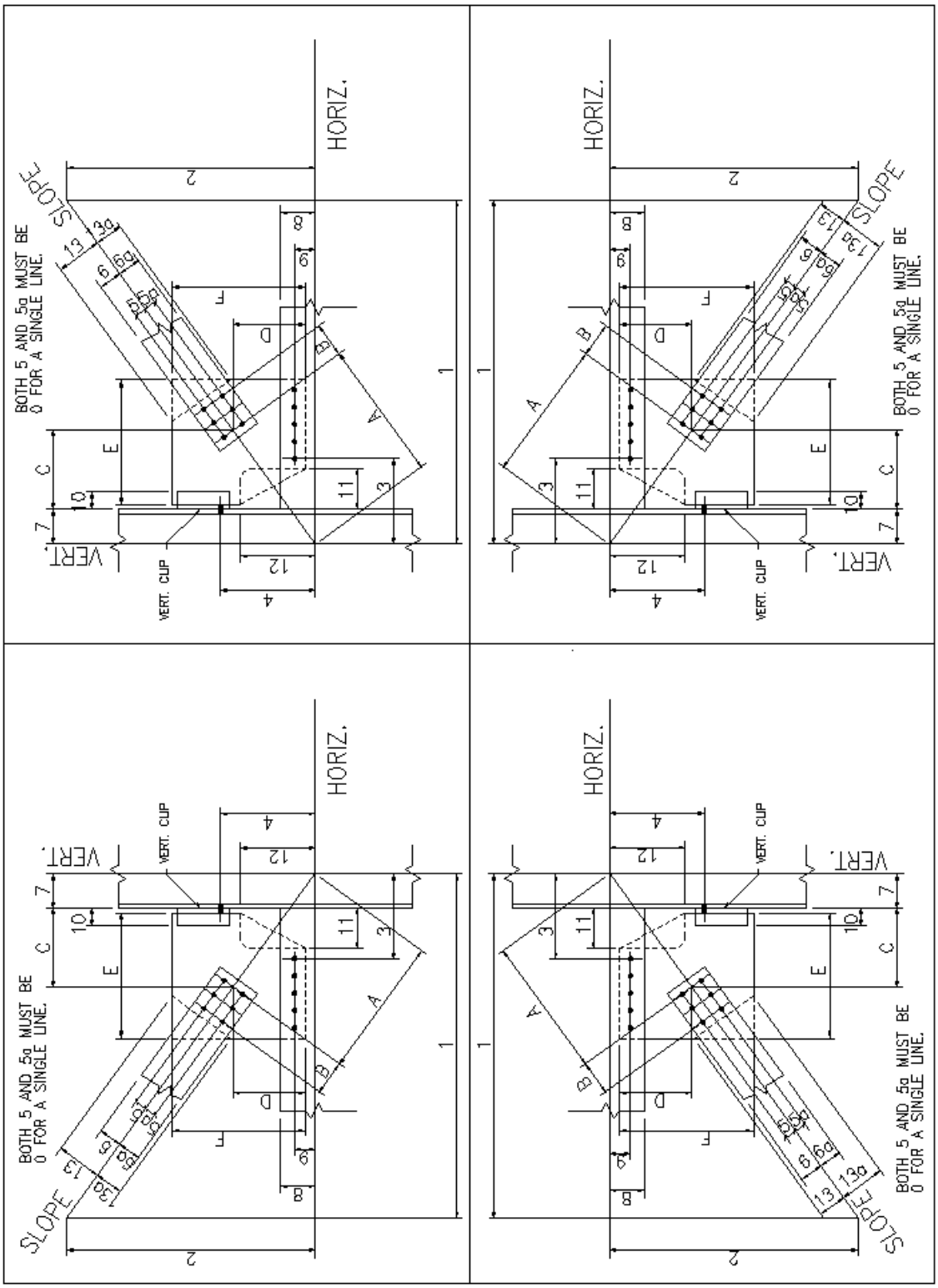

HORIZ.
VERT.
SLOPE
BOTH 5 AND 5a MUST BE 0 FOR A SINGLE LINE.
TYPE 2


HORIZ.
VERT.
SLOPE
BOTH 5 AND 5a MUST BE
0 FOR A SINGLE LINE.
TYPE 3


BOTH 5 AND 5a MUST BE
0 FOR A SINGLE LINE.
HORIZ
VERT
SLOPE
TYPE 4


HORIZ.
VERT.
SLOPE
HORIZ. CLIP
BOTH 5 AND 5a MUST BE 0 FOR A SINGLE LINE.

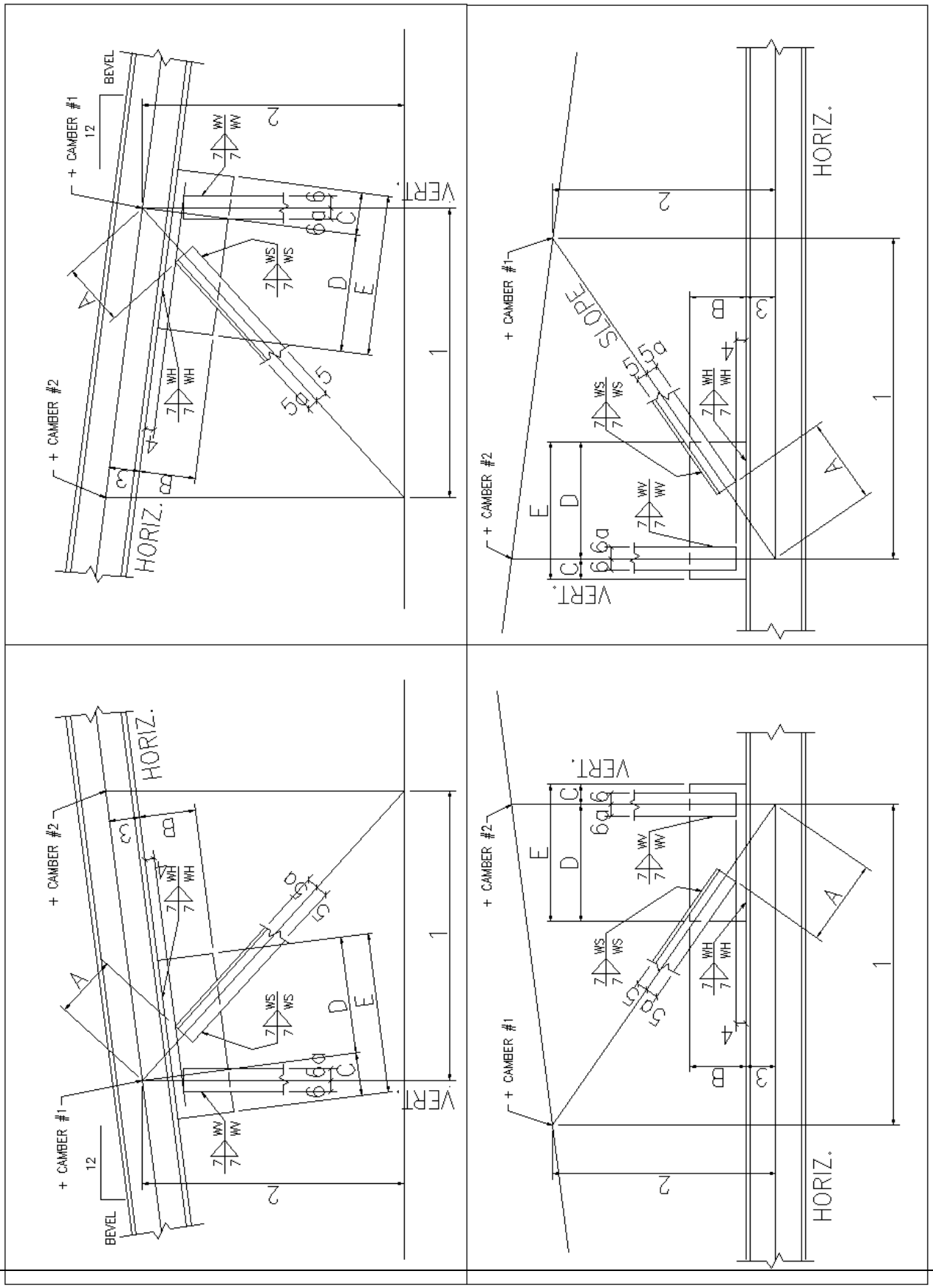
TYPE 5

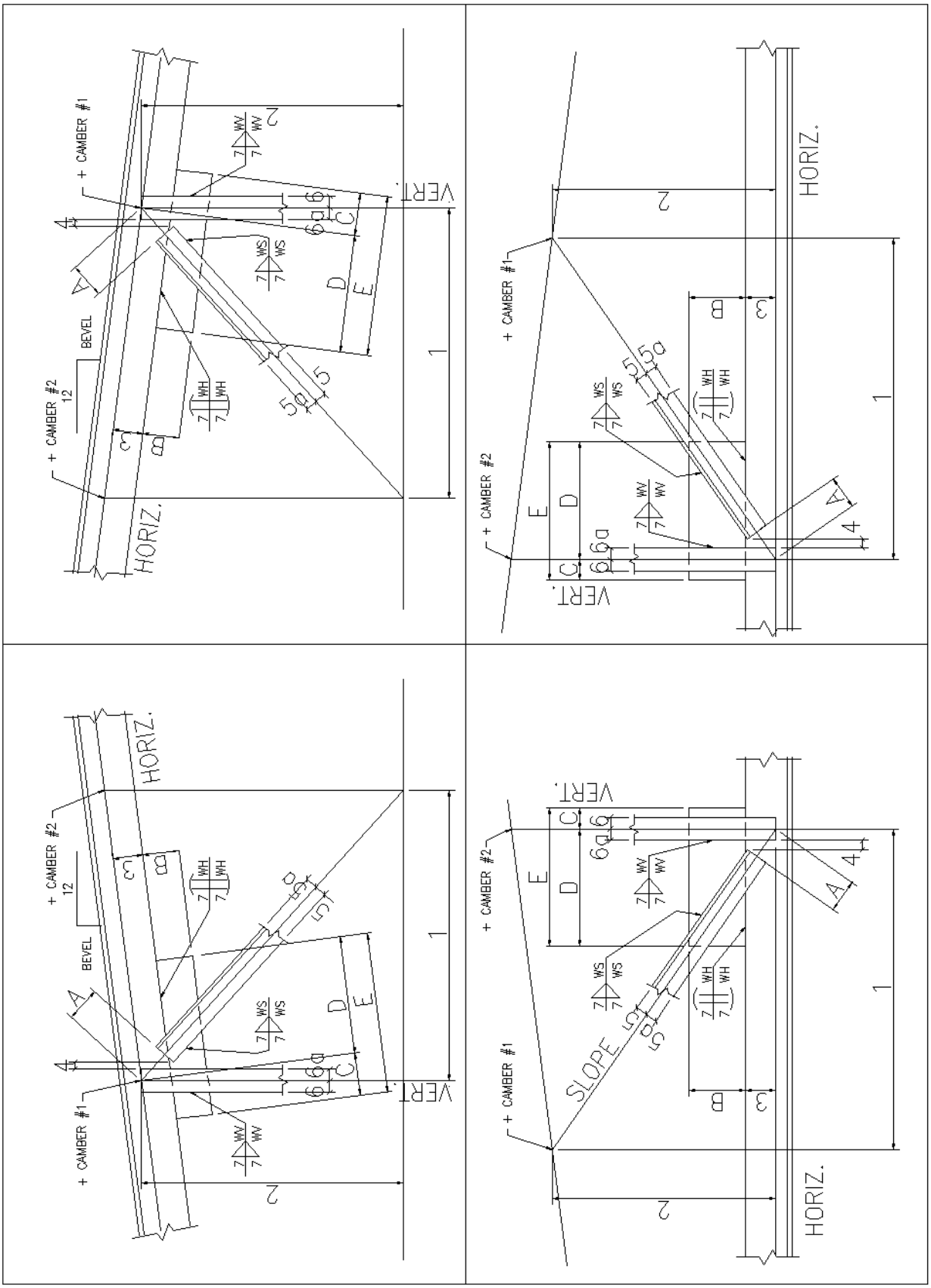

HORIZ.
VERT.
SLOPE
VERT. CUP
BOTH 5 AND 5a MUST BE 0 FOR A SINGLE LINE.

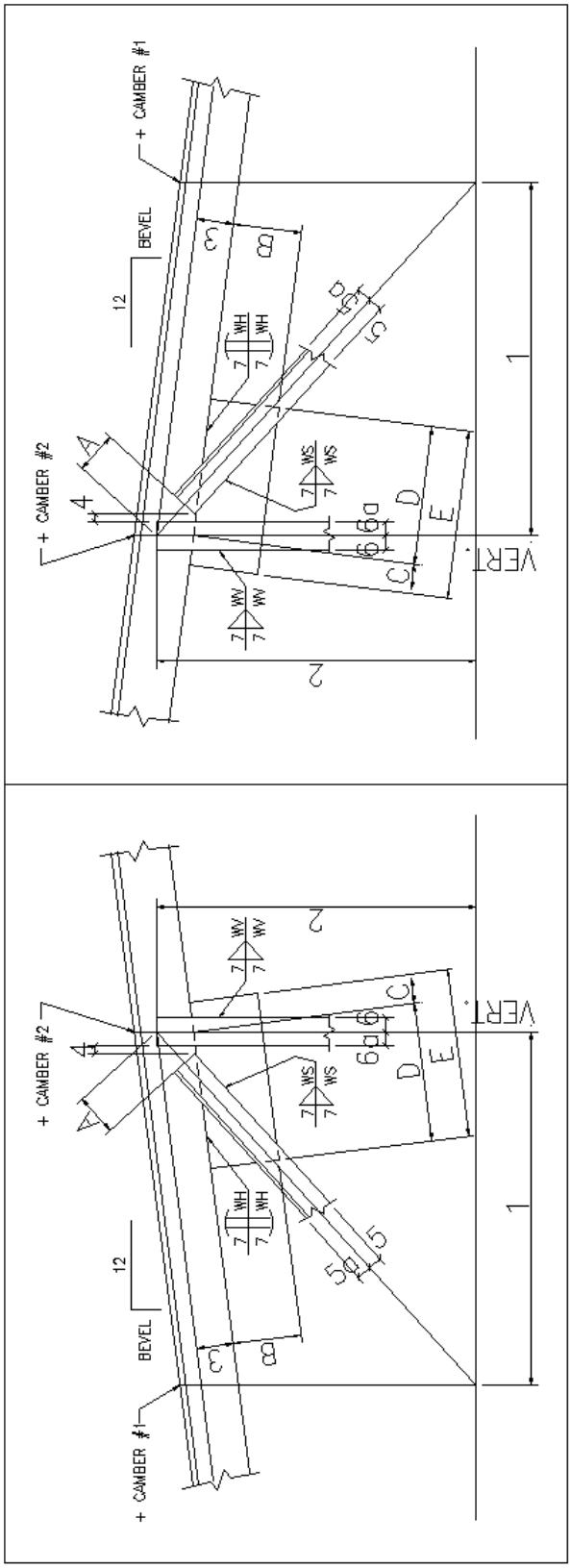
TYPE 6

Appendix D – Truss Panel Details

This appendix provides you with a copy of the slides that are used with the GUSSET PLATE (TRUSS) program.


+ CAMBER #1
+ CAMBER #2
BEVEL
12
HORIZ.
VERT.
SLOPE
A
B
C
D
E
1
2
3
4
5a
5b
6a
6b
6c
6d
6e
6f
6g
WV
WH
WS


+ CAMBER #1
+ CAMBER #2
12
BEVEL
HORIZ.
VERT.
SLOPE
WV
WS
WH
A
B
C
D
E
1
2
3
4
5a
5b
6a
6b
6c
6d
6e
6f
6g


+ CAMBER #1
BEVEL
12
+ CAMBER #2
VERT.
WH
WS
WV

Appendix E – Stair Input Forms

The following forms in this appendix can be used for entering data into the stair program. The forms can be filled out by someone that does not have access to AutoSD Steel Detailing and later entered by someone who does. You may make copies of the forms for your use. For the picture forms on page 250 and 252, put an X in the boxes provided to select the top and bottom condition type and other options available. Fill in the boxes for the dimensions. Some boxes will be used based on which end conditions were selected. Answer the questions on the form on page 250.

If bottom is type 1, 2, 3, 4, 6 or 7 with clip or plate

Sub assembly mark for bottom connection _____
Clip is (1)NS, (2)FS, (3)BS .....
Gage of clip .....
Add piece to shop bill …….Yes _____No _____
Quantity .....
Description _____
Sub assembly mark for bottom closure plate _____
Add piece to shop bill ..... Yes _____ No ____
Quantity .....

If bottom is type 3 or 7

Minus dimension at lower end ..... _____
Sub assembly mark for bottom stringer _____
Are marks RIGHT and LEFT …….Yes ____ No ____
Add piece to shop bill …….. Yes ____ No ____
Quantity ..... _____

If top is with clip or end plate

Sub assembly mark .....
Clip is (1)NS, (2)FS, (3)BS .....
Gage of clip or end plate .....
Add piece to shop bill .....Yes _____No _____
Quantity .....
Description _____

If #6 is negative

Sub assembly mark for top vertical closure plate_____
Add piece to shop bill …….Yes _____ No _____
Quantity ..... _____

If top is type 1, 3 or 4

Assembly mark for top horizontal closure plate _____
Add piece to shop bill …….Yes _____ No _____
Quantity.....

If top is type 2

Sub assembly mark for top stringer .....
Are marks RIGHT and LEFT …… Yes _____No _____
Add piece to shop bill ..... Yes _____No _____
Quantity ..... _____
Sub assembly mark for sloping stringer ._____
Are marks RIGHT and LEFT …… Yes ____ No _____
Add piece to shop bill .....Yes ____ No _____
Quantity ..... _____
Tread type (Pan/Grating) ……... P _____G _____

If Grating Treads

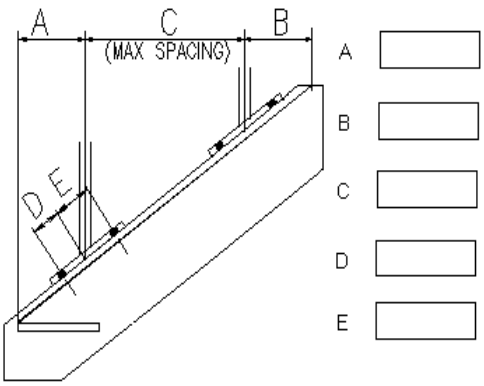
Horiz. dim. from W.P. to hole <1 1/8">: _____
Horiz. dim. from hole to hole <7">: .....
Vert. dim. from W.P. to hole <1 3/4">: ..

If PanTreads See page **177**

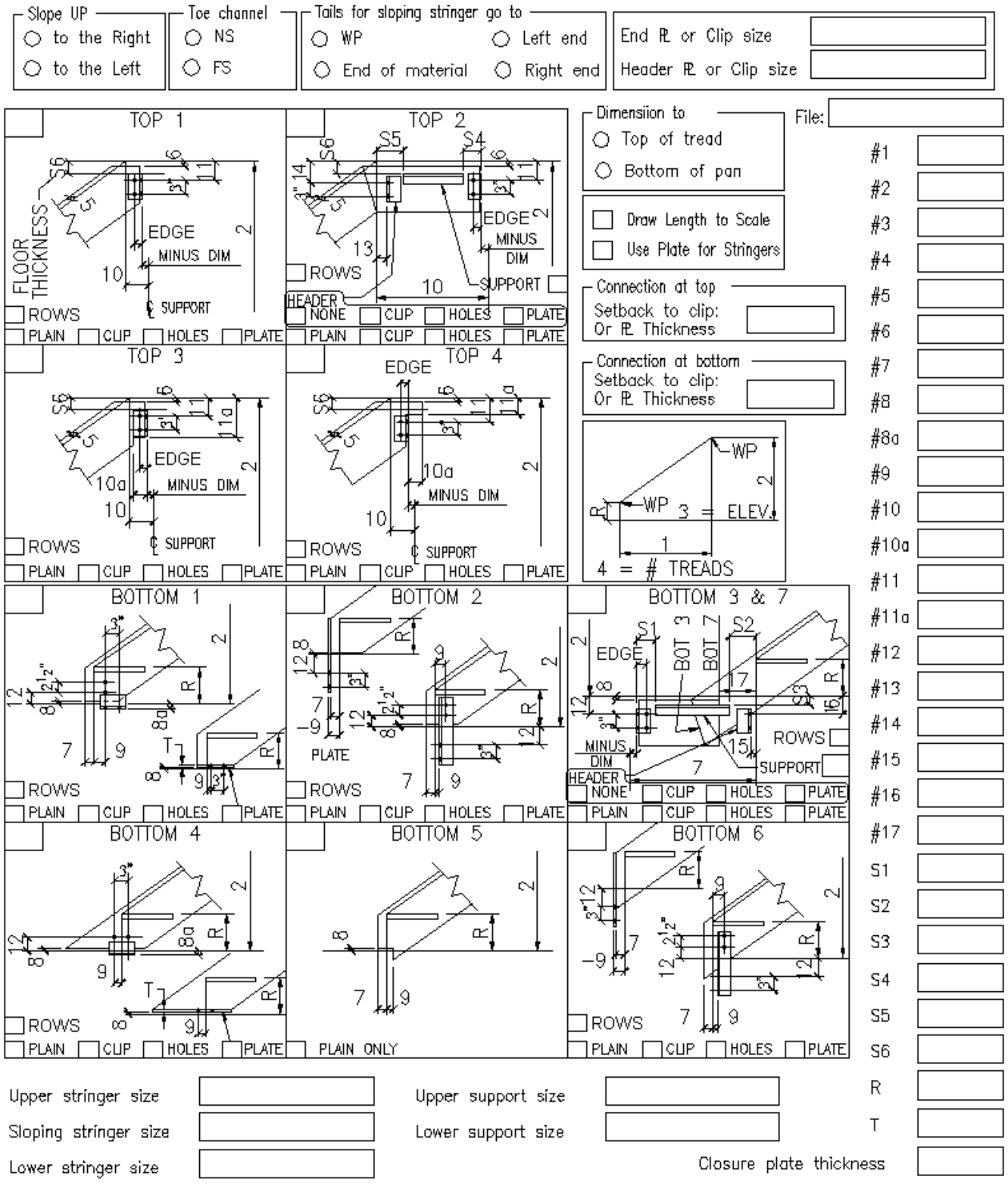
Standard Tread name .....
First Bottom Tread name ... _____
Are tail dimensions required …. Yes _____ No _____

Minus dimension at upper end ..... _____

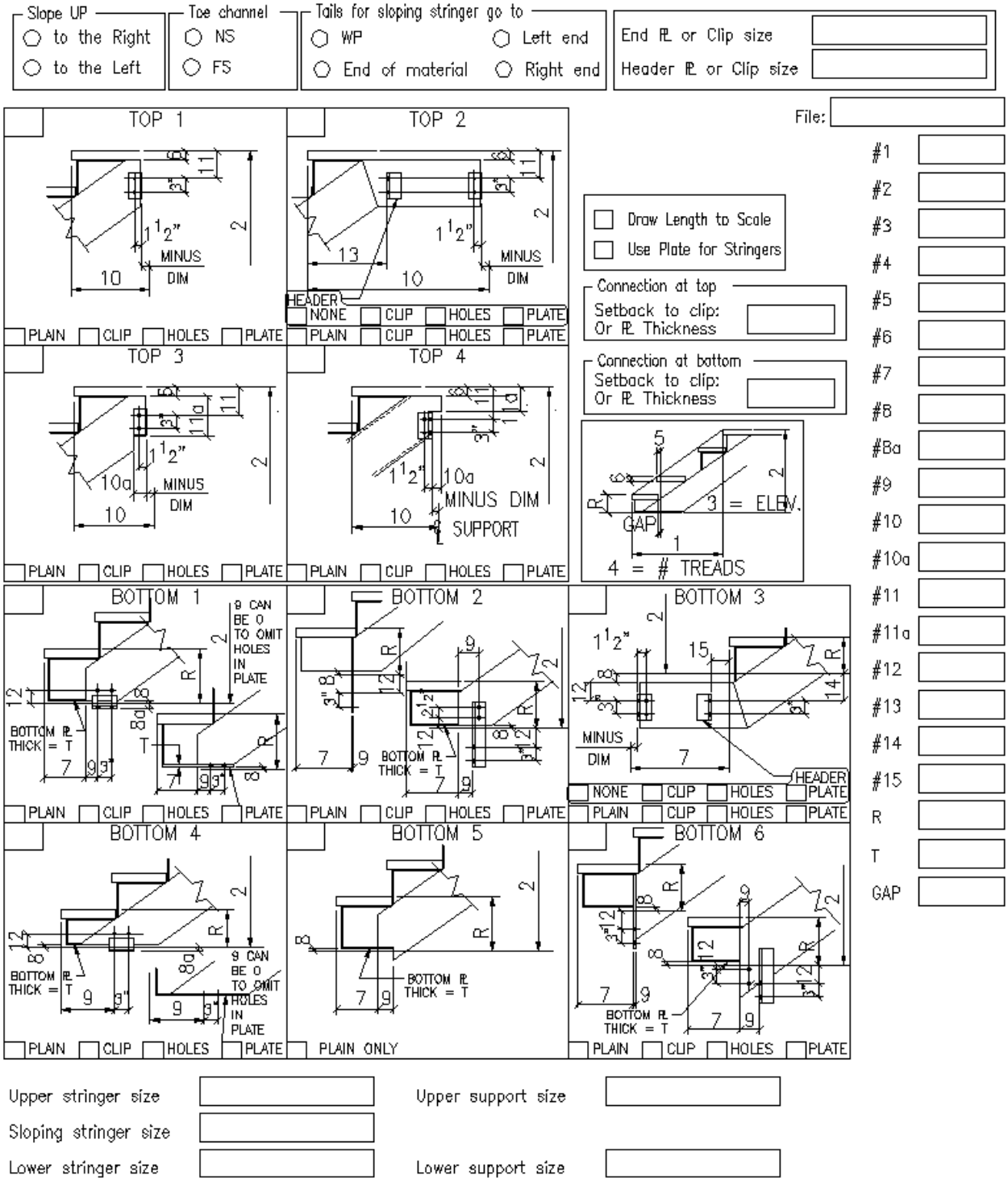
Include Handrail holes ……..Yes _____ No _____



Standard stair with pan or grating treads.


Slope UP
to the Right
to the Left
Toe channel
NS
FS
Tails for sloping stringer go to
WP
Left end
End of material
Right end
End PL or Clip size
Header PL or Clip size
TOP 1
TOP 2
TOP 3
TOP 4
FLOOR THICKNESS
EDGE
MINUS DIM
SUPPORT
ROWS
HEADER
PLAIN
CLIP
HOLES
PLATE
NONE
Dimension to
Top of tread
Bottom of pan
Draw Length to Scale
Use Plate for Stringers
Connection at top
Setback to clip:
Or PL Thickness
Connection at bottom
Setback to clip:
Or PL Thickness
WP
WP 3 = ELEV.
4 = # TREADS
File:
#1
#2
#3
#4
#5
#6
#7
#8
#8a
#9
#10
#10a
#11
#11a
#12
#13
#14
#15
#16
#17
S1
S2
S3
S4
S5
S6
R
T
BOTTOM 1
BOTTOM 2
BOTTOM 3 & 7
BOTTOM 4
BOTTOM 5
BOTTOM 6
PLATE
PLAIN ONLY
Upper stringer size
Sloping stringer size
Lower stringer size
Upper support size
Lower support size
Closure plate thickness

"Butterfly" stair with pre-cast treads on top of stringer



Thickness of tread support Flat Bars ..... _____

Width of tread support Flat Bars ..... _____

Sub assembly mark for Bottom tread support ..... _____

Sub assembly mark for Standard tread supports ... _____

ERROR HANDLING AND REPORTING

AutoSD, Inc. takes every step to insure the quality and performance of the products it sells. There are however, times when a certain course of events that we have not subjected the software may cause an error. If you experience problems we want to hear about them. If you have a suggestion or an idea of how the product can better serve your needs we encourage you to contact us. The future of AutoSD depends on the comments and needs of our users.

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Address: _____

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____ AutoCAD _____ BricsCAD, Version _____

Operating system:

Windows 98 _____ NT _____ 2000 _____ XP _____ Vista _____ 7 _____ 8 _____

State the problem:

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